3D/2D modelling suite for integral water solutions

DELFT3D

Deltares systems

RGFGRID

Deltares

Enabling Delta Life

User Manual

RGFGRID

Generation and manipulation of curvilinear grids for Delft3D-FLOW and Delft3D-WAVE

User Manual

Hydro-Morphodynamics

Version: 4.00.34074

26 May 2014

RGFGRID, User Manual

Published and printed by:
Deltares
Boussinesqweg 1
2629 HV Delft
P.O. Box 177
2600 MH Delft
The Netherlands

telephone: +31 88 335 82 73
fax: +31 88 335 85 82
e-mail: info@deltares.nl
www: http://www.deltares.nl

For sales contact:
telephone: +31 88 335 81 88
fax: +31 88 335 81 11
e-mail: sales@deltaressystems.nl
www: http://www.deltaressystems.nl

For support contact:
telephone: +31 88 335 81 00
fax: +31 88 335 81 11
e-mail: support@deltaressystems.nl
www: http://www.deltaressystems.nl

Contents

List of Figures

List of Tables

1 Guide to this manual

1.1 Introduction

This User Manual concerns the grid generation module, RGFGRID, of the Delft3D software suite. To make this manual more accessible we will briefly describe the contents of each chapter and appendix.

If this is your first time to start working with RGFGRID module we suggest you to read and practice the getting started of Chapter 3 and the tutorial of Chapter 6. These chapters explain the user interface options and guide you through the generation of your first grid.

Chapter 2: Introduction to RGFGRID , provides specifications of RGFGRID and the areas of applications.

Chapter 3: Getting started, explains the use of the overall menu program, which gives access to all Delft3D modules and to the pre- and post-processing tools. Last but not least you will get a first introduction into the RGFGRID Graphical User Interface, used to define a grid which can be used in a hydrodynamic or wave simulation.

Chapter 4: General operation, provides practical information on the general operation of the RGFGRID module.

Chapter 5: Menu options, provides a description of all menu and toolbar options.

Chapter 6: Tutorial, emphasis at giving you some first hands-on experience in using the RGFGRID module to define the input of a simple problem and in executing a water quality simulation.

References, provides a list of publications and related material on the RGFGRID module.

Appendix A: Files of RGFGRID , gives a description of the files that can be used in RGFGRID as input or output. Generally, these files are generated by RGFGRID or other modules of the Delft3D suite and you need not to be concerned about their internal details. However, in certain cases it can be useful to know these details, for instance to generate them by means of other utility programs.

1.2 Name and specifications of the program

Title	RGFGRID
Description	RGFGRID is a program for generation and manipulation of curvilinear grids for Delft3D-FLOW and Delft3D-WAVE. The co-ordinate system may be Cartesian or spherical. Delft3D-FLOW is a simulation program for hydrodynamic flows and transports in 2 and 3 dimensions, see Delft3D-FLOW (2013). One of the wave modules in Delft3D-WAVE is SWAN which is a wave energy simulation package; see SWAN (2000).
Special facilities	sketch of coarse grid using splines smooth refinement module orthogonalisation module various grid manipulation options grid design by bathymetry or polygon control Cartesian or spherical co-ordinates dynamic memory allocation multiple grids supported

1.3 Manual version and revisions

A manual applies to a certain release of the related numerical program. This manual applies to RGFGRID, version 4.20.

1.4 Typographical conventions

Throughout this manual, the following conventions help you to distinguish between different elements of text to help you learn about RGFGRID.

Example	**Description**
Waves **Boundaries**	Title of a window or sub-window. Sub-windows are displayed in the **Module** window and cannot be moved. Windows can be moved independently from the **Module** window, such as the **Visualisation Area** window.
Save	Item from a menu, title of a push button or the name of a user interface input field. Upon selecting this item (click or in some cases double click with the left mouse button on it) a related action will be executed; in most cases it will result in displaying some other (sub-)window. In case of an input field you are supposed to enter input data of the required format and in the required domain.
<\tutorial\wave\swan-curvi> <siu.mdw>	Directory names, filenames, and path names are expressed between angle brackets, <>. For the Linux and UNIX environment a forward slash (/) is used instead of the backward slash (\) for PCs.

Example	Description
“27 08 1999”	Data to be typed by you into the input fields are displayed between double quotes. Selections of menu items, option boxes etc. are described as such: for instance ‘select *Save* and go to the next window’.
`delft3d-menu`	Commands to be typed by you are given in the font Courier New, 10 points.
	User actions are indicated with this arrow.
[m/s] [-]	Units are given between square brackets when used next to the formulae. Leaving them out might result in misinterpretation.

1.5 Changes with respect to previous versions

Version	Description
4.00.00	Complete new version of RGFGRID

2 Introduction to RGFGRID

2.1 Introduction

The purpose of the RGFGRID program is to create, modify and visualise orthogonal, curvilinear grids for the Delft3D-FLOW module.

Curvilinear grids are applied in finite difference models to provide a high grid resolution in the area of interest and a low resolution elsewhere, thus saving computational effort.

Grid lines may be curved along land boundaries and channels, so that the notorious 'stair case' boundaries, that may induce artificial diffusion, can be avoided.

Curvilinear grids should be smooth in order to minimise errors in the finite difference approximations. Finally, curvilinear grids for Delft3D-FLOW have to be orthogonal, which saves some computationally expensive transformation terms. Extra effort in the model set-up phase, results in faster and more accurate computations.

2.2 Co-ordinate systems

The grid system used in RGFGRID can be either Cartesian (in metres) or spherical (in decimal degrees). Cartesian co-ordinates can be displayed on a screen directly, just using a scale factor. Spherical co-ordinates can be displayed on screen as plane co-ordinates or as projected co-ordinates. Plane co-ordinates on screen give distortion in the polar direction. Depending on the type of projection, projected co-ordinates have no distortion in distance and angles. For this reason a stereographic projection is used in RGFGRID.

Starting from scratch, you have to select a co-ordinate system. The co-ordinates of all objects (land boundary, splines, grid, samples, etc.) are then in the selected co-ordinate system. When opening a grid, RGFGRID will read the co-ordinate system of the imported grid. The co-ordinates of other objects (land boundary, splines, polygons, samples and text files) are not checked; this is the responsibility of the user.

2.3 Program considerations

RGFGRID is designed to create grids with minimum effort, fulfilling the requirements of smoothness and orthogonality. The program allows for an iterative grid generation process, starting with a rough sketch of the grid by splines. Then, the splines are transformed into a grid, which can be smoothly refined by the program. Whenever necessary, you can orthogonalise the grid in order to fulfil the Delft3D-FLOW requirement of orthogonality.

Various grid manipulation options are provided in order to put the grid lines in the right position with the right resolution. For instance, a grid line can be 'snapped' to a land boundary. The surrounding grid smoothly follows. More detail is brought into the grid after every refinement step.

Existing grids may be modified or extended using this program. Grids can be locally refined by insertion of grid lines. The resulting local 'jump' in grid sizes can be smoothed by a so-called 'line smoothing'.

Bathymetry data can be displayed on the screen, so that internal gullies can be taken into account while drawing the design grid. Existing model grids can be opened and displayed on the screen, while creating new grids to be pasted later to the original. Before each modification or edit action, the grid is saved to the so-called 'previous grid'. Pressing `Esc` after an edit action, copies the previous grid back to the grid. If desired, the previous grid can be shown together with the active grid.

Grid properties such as smoothness, resolution, orthogonality etc, can be visualised to check the grid quality. Graphical output can easily be created in various formats.

3 Getting started

3.1 Overview of Delft3D

The Delft3D program suite is composed of a set of modules (components) each of which covers a certain range of aspects of a research or engineering problem. Each module can be executed independently or in combination with one or more other modules.

Delft3D is provided with a menu shell through which you can access the various modules. In this chapter we will guide you through some of the input screens to get the look-and-feel of the program. In the Tutorial, Chapter 6, you will learn to define a simple scenario.

3.2 Starting Delft3D

To start Delft3D:

- ◇ On an MS Windows platform: select *Delft3D* in the *Programs* menu.
- ◇ On Linux machines: type `delft3d-menu` on the command line.

Next the title window of Delft3D is displayed, Figure 3.1.

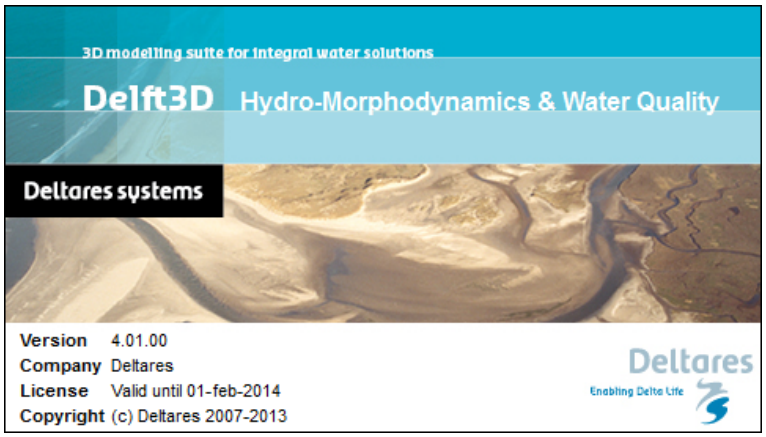


Figure 3.1: *Title window of Delft3D*

After a short while the main window of the Delft3D-MENU appears, Figure 3.2.

Several menu options are shown. For now, only concentrate on exiting Delft3D-MENU, hence:

- ◇ Click on the *Exit* push button.

The window will be closed and you are back in the Windows Desktop screen for PCs or on the command line for Linux workstations.

Remark:

- ◇ In this and the following chapters several windows are shown to illustrate the presentation of Delft3D-MENU and RGFGRID. These windows are grabbed from the PC-platform. For Linux workstation the content of the windows is the same, but the colours may be different.

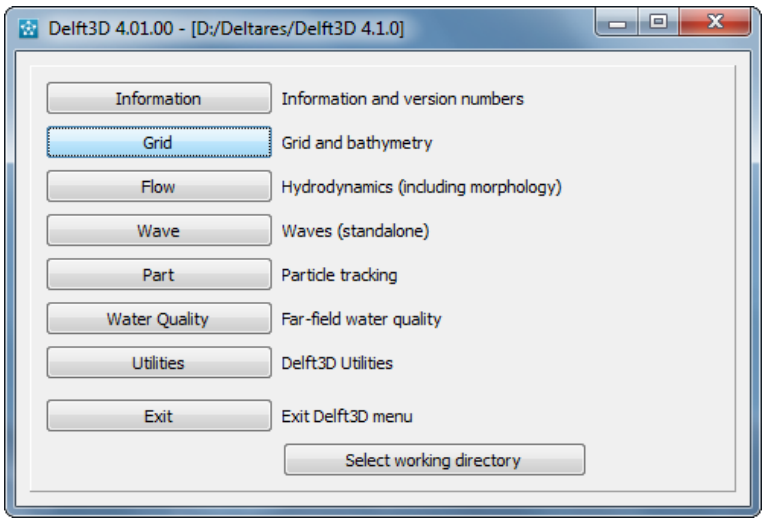


Figure 3.2: *Main window* ***Delft3D-MENU***

3.3 Getting into RGFGRID

To continue start the menu program again as indicated in Section 3.2.

◇ Click the *Grid* button, see Figure 3.2

Next the selection window for **Grid and bathymetry** is displayed for preparing a curvilinear grid, interpolate data on that grid and aggregate the hydrodynamic cells, see Figure 3.3.

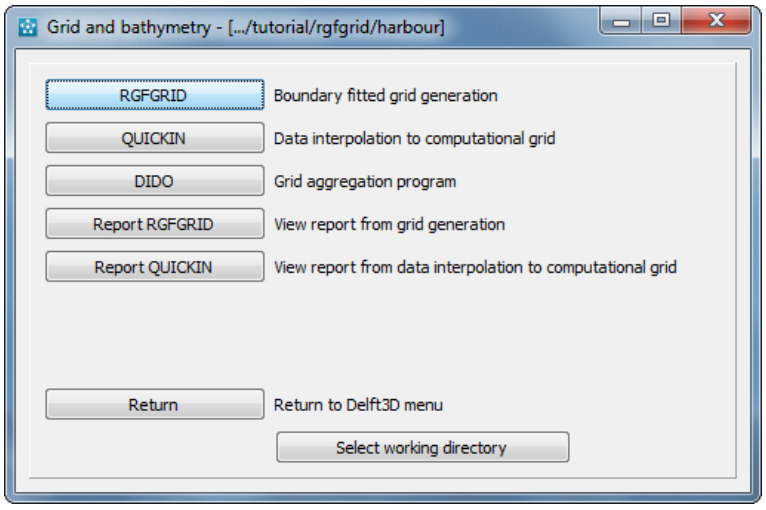


Figure 3.3: *Selection window for* ***Grid and Bathymetry***

Note that in the title bar the current directory is displayed, in our case <D:/delft3d>.

Before continuing with any of the selections of this **Grid and bathymetry** window, you select the directory in which you are going to prepare scenarios and execute computations:

◇ Click the *Select working directory* button.

Next the **Select working directory** window is displayed, see Figure 3.4 (your current directory may differ, depending on the location of your Delft3D installation).

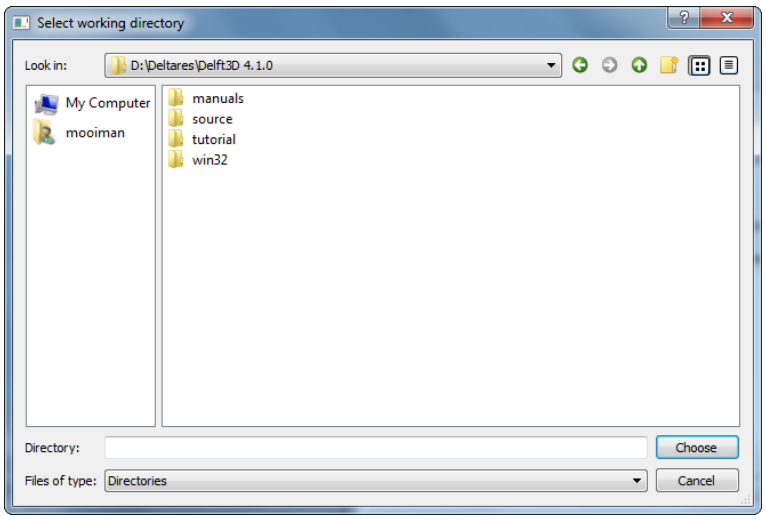


***Figure 3.4: Select working directory** window*

- ◇ Browse to and open the <tutorial> sub-directory of your Delft3D Home-directory.
- ◇ Open the <rgfgrid> directory.
- ◇ Open the <harbour> directory.
- ◇ Close the **Select working directory** window by clicking button *Choose*, see Figure 3.5.

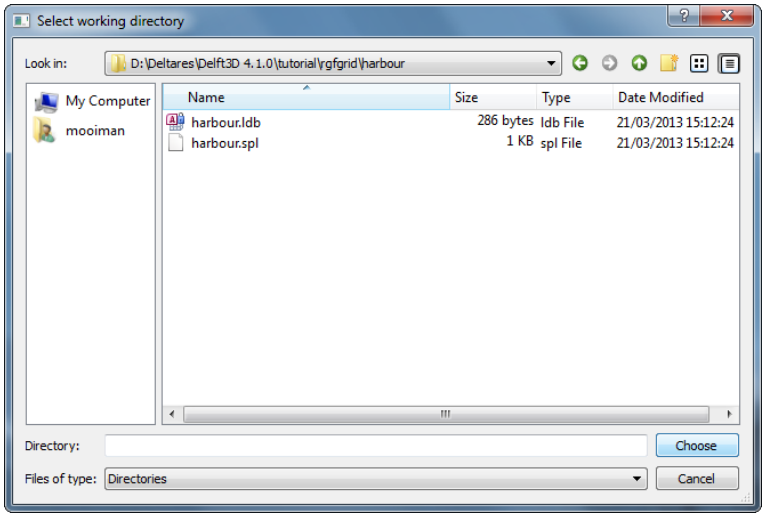


***Figure 3.5: Select working directory** window to set the working directory to <rgfgrid/habour>*

Next the **Grid and bathymetry** window is re-displayed, but now the changed current working directory is displayed in the title bar, see Figure 3.6.

***Figure 3.6:** A part of the current working directory is shown in the title bar due to its length*

Remark:

- ◇ In case you want to start a new project for which no directory exists yet, you can select  in the **Select working directory** window to create a new folder.
- ◇ Click on *RGFGRDID* in the **Grid and bathymetry** window, see Figure 3.3.

RGFGRID is loaded and the primary input screen is opened, Figure 3.7.

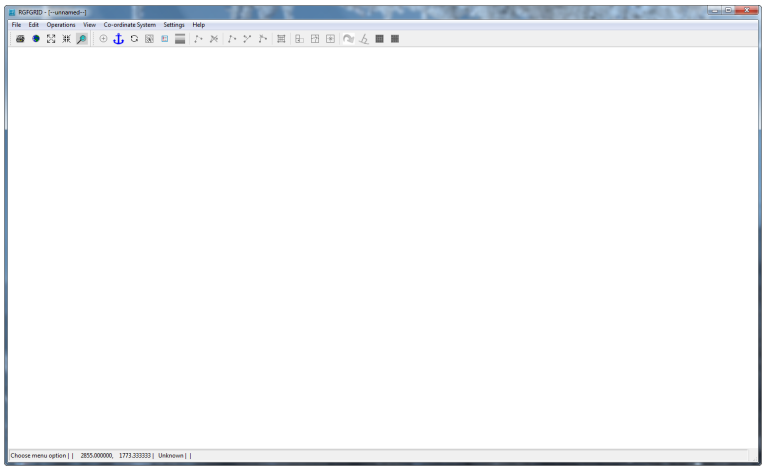


Figure 3.7: *Main window of the RGFGRID*

In the lower-left corner of the status bar RGFGRID gives additional operational information, see Figure 3.8, such as:

- ◇ User selections.
- ◇ Operational instructions (for instance Toggle anchor mode).
- ◇ x and y co-ordinates of the current cursor position.
- ◇ Co-ordinate system: Cartesian or Spherical.
- ◇ Distance (in metre) to a user-defined anchor point (only displayed when the anchor is activated).

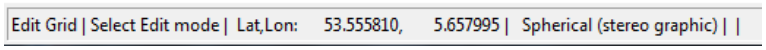


Figure 3.8: *Operational information displayed in the statusbar*

3.4 Exploring some menu options

First, set the co-ordinate system to the system you want to work in. Since we are going to work in the Cartesian co-ordinate system:

- ◇ On the *Co-ordinate System* menu click *Cartesian Co-ordinates*, see Figure 3.9

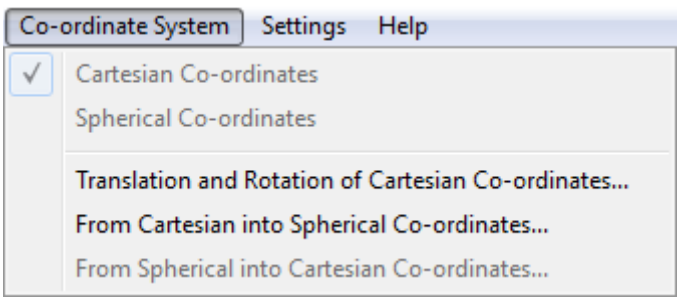


Figure 3.9: Co-ordinate System *menu, Cartesian Co-ordinates selected*

To open a land boundary:

◇ Upon selecting *File* → *Attribute Files* → *Open Land Boundary*, you can open a collection of land boundaries, see Figure 3.10. Land boundaries (or land-water marking) are in files with default mask <∗.ldb>.

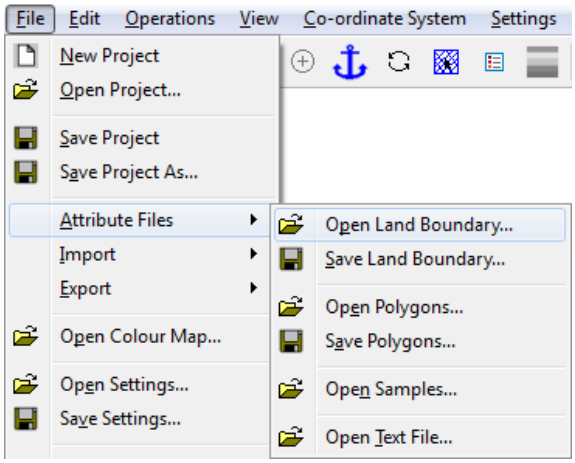


Figure 3.10: *Menu item* File → Attribute Files → Open Land Boundary

Next the **Open Land Boundary** window is displayed, see Figure 3.11.

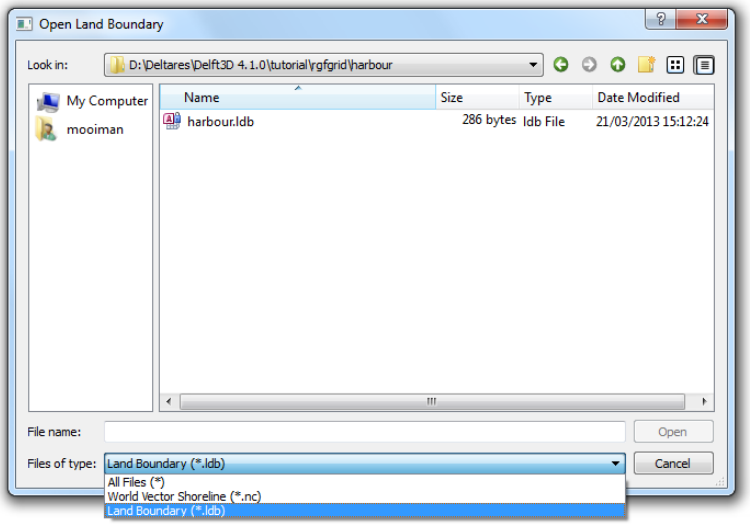


Figure 3.11: *File open window* ***Open Land Boundary***

In the current directory one land boundary file is present.

◇ Select <harbour.ldb> and click *Open* to open the land boundary file.
◇ On the *Edit* menu point to *Spline* and click *New*, or click on the toolbar.
◇ To draw a spline, click with the left-mouse to define spline-points. To finish the current spline click with the right-mouse. Click left to start with the next spline. The result may look like as in Figure 3.12

Practise with zooming in or out. To zoom in or out, either:

◇ Click on to zoom in and zoom out on the toolbar.
◇ Press the + and - key while keeping the CTRL-key pressed.

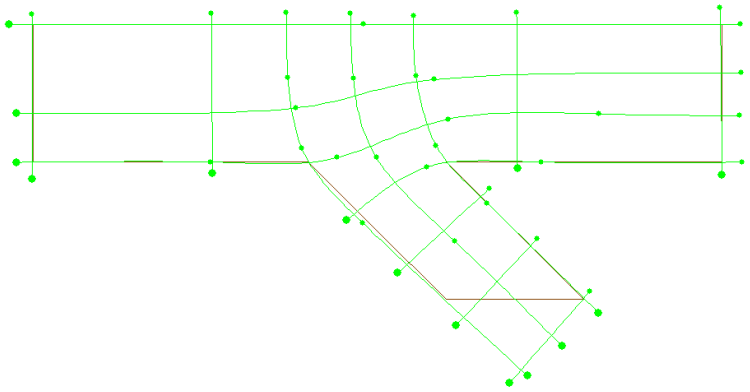


Figure 3.12: *Example of a spline grid*

- ◇ Use the mouse scroll wheel.
- ◇ To delete an entire spline, click on the toolbar and click one of the supporting points of the spline to be deleted.
- ◇ To delete a single point of a spline, click and click a spline point to delete this single point.
- ◇ To move a single point of a spline, click or press R, click the point and click again at the new location.

Now we delete this spline grid:

- ◇ On the *Operations* menu, point to *Delete* and click *Splines*, see Figure 3.13

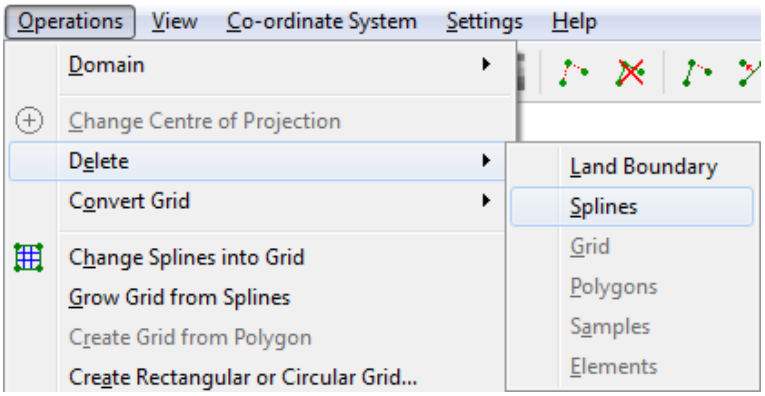


Figure 3.13: *Menu option* Operations → Delete → Splines

We will continue with an existing splines file

- ◇ On the *File* menu, point to *Import* and click *Splines*.
- ◇ Select <harbour2.spl>. After selection the file is loaded and displayed, see Figure 3.14.

- ◇ On the *Operations* → *Change Splines into Grid*, or click on the toolbar.

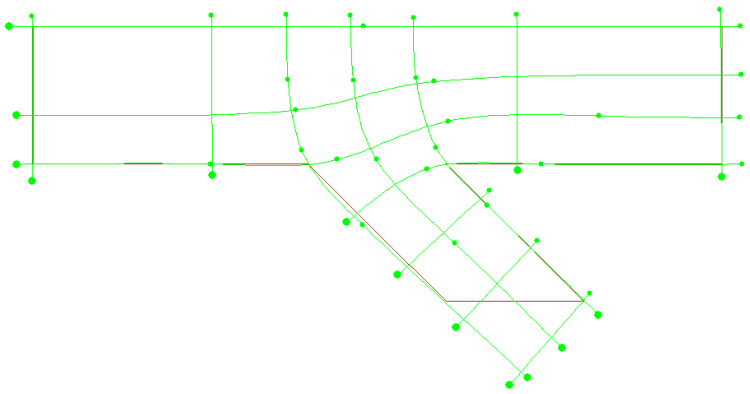


Figure 3.14: *Spline grid from tutorial file <harbour.spl>*

This operations transforms the spline grid into a grid and at the same time refines it 3 times in both directions, see Figure 3.15. The refinement factors can be set in the **General Parameters** form menu item *Settings* → *General*.

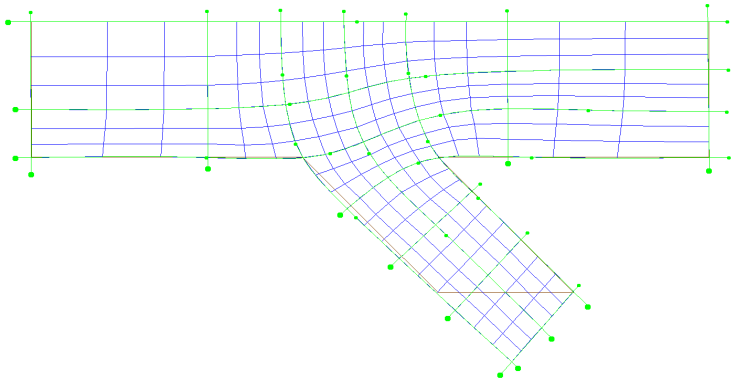


Figure 3.15: *Result of operation* OPerations → Change Splines into Grid

To save the grid

◇ On the *File* menu, point to *Export* and click *Grid*

The **Save As** window opens, see Figure 3.16.

◇ Type <harbour> and click *Save* to save your grid

You will be back in the main window of RGFGRID.

3.5 Exiting RGFGRID

To exit the RGFGRID

◇ Click *Exit* on the *File* menu.

You will be back in the **Grid and bathymetry** window, see Figure 3.3

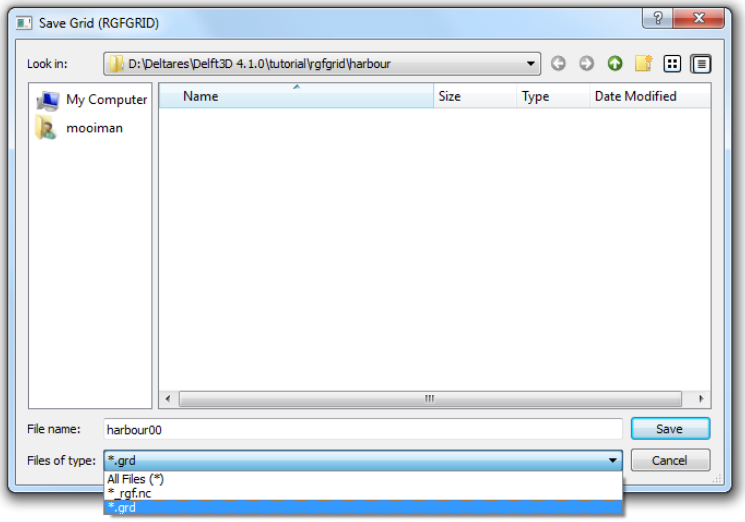


Figure 3.16: *Window* ***Save Grid*** *to save grid file*

- ◇ Click *Return* to return to the main window of Delft3D-MENU, see Figure 3.2
- ◇ Click *Exit*.

The window is closed and the control is returned to the desk top or the command line.

In this Getting Started session you have learned to access the RGFGRID and to open and and to generate and save a grid file.

We encourage new users next to run the tutorial described in Chapter 6.

4 General operation

4.1 General program operation instruction

The main menu bar is positioned at the top of the screen.

Help

Upon selecting *Help* → *User Manual*, the RGFGRID User Manual in PDF-format will be opened. Use the bookmarks in the contents to locate the subject you are interested in.

Print screen

Press `Ctrl-P` or click  on the toolbar to obtain the print window for a hardcopy of the current screen.

File menu

The file-menu is the standard **Open** and **Save As** window. The file mask depends on the type of data that you want to open or save. You can change the directory by navigating through the folders.

It is possible to specify whether to Stay on the Start-up Directory or not, in the **Settings General** form.

General cursor and keyboard functions

The left mouse button activates or confirms desired actions. The `Esc` key cancels the last edit action. The right mouse button may also confirm actions, or may put the program back into its original mode.

4.2 Key stroke functions

Key `A` = Anchor, or on toolbar 
When clicking  on the toolbar and next pressing the `A` key on the keyboard, a so-called anchor will appear, which acts as zero-distance point. The distance (in metre) of the present cursor position to this point is displayed in the status bar at the right of the co-ordinate system indicator, see Figure 4.1. Moving the cursor around and pressing `A` again will relocate the anchor. Clicking  again will de-activate the anchor.

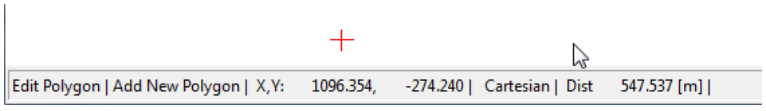


Figure 4.1: *Location of anchor + and distance between anchor and cursor at the right*

Key `D` = Delete
In the *Edit* → *Polygon* options, pressing `D` allows you to delete individual points (polygon, depth or sample).

Key `E` = Erase polygon
In *Edit* → *Polygon*, keeping `E` pressed allows you to delete the indicated polygon.

Key `I` = Insert
In *Edit* → *Polygon*, pressing `I` starts the vertex insert action depending on the first click on the screen, there are two options.

Normal	If the first click is in between two vertices of the polygon then a point will be inserted in the closest edge.
Incremental	If the first click is on a vertex of the existing polygon the the incremental insert option is started, which means all next locations click by the left mouse button are added to the polygon behind the indicated point. This mode will be cancelled when indication a existing point of the polygon and the normal insert option will be used. Hitting another key will also cancel this insert option.

Key `Ctrl-P` = Print screen
Pressing `Ctrl-P` will open the print window. The current screen will be printed to your printer or to a file.

Key `R` = Replace
In *Edit* → *Polygon*, pressing `R` allows you to replace (move) individual points.

Key `Mouse wheel`
Use the mouse wheel to zoom in and zoom out. Other ways are:

Click  on the toolbar to zoom in.

Click  on the toolbar to zoom out.

To define a zoom box, click  on the toolbar and drag a box.

Key `Ctrl +` = Zoom in
Keep the `Ctrl`-key pressed and use the + key to zoom in more.

Key `Ctrl -` = Zoom out
Keep the `Ctrl`-key pressed and use the - key to zoom in more.

Key `Ctrl` move cursor = move focus of screen
Keep the `Ctrl`-key pressed and move the cursor around. The current screen will move accordingly.

Key `Ctrl` arrow keys = move focus of screen left, right, up or down
Keep the `Ctrl`-key pressed and use the arrow keys to move the focus of the screen accordingly.

Key `Esc` = Undo
In various edit modes the latest action will be undone pressing `Esc`.

5 Menu options

The menu bar contains the following items, see Figure 5.1, each item is discussed in a separate section.

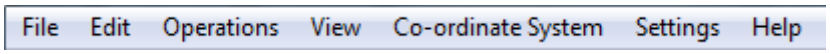


Figure 5.1: *RGFGRID menu options*

5.1 File menu

Before opening an object (land boundary, polygon, samples or splines) be sure you have set the co-ordinate system on the *Co-ordinate System* menu, see Section 5.5.

- ◇ When opening files, RGFGRID will not check the co-ordinate system in the file against the current co-ordinate system in RGFGRID, except when opening a grid.

On the *File* menu, see Figure 5.2, options are available to open a project (collection of grids and ddb-file), attribute files required for the definition of a grid (i.e. land boundary and samples) and to import grid related files (grids, splines and DD boundaries). The results at each stage of the grid definition process can be saved. The option to quit RGFGRID is located here also.

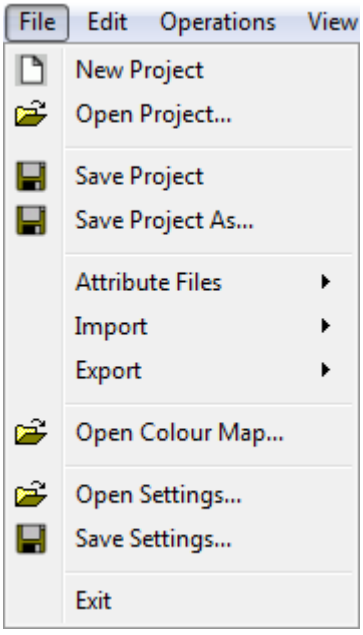


Figure 5.2: *Options on the* File *menu*

The start-up directory to open and save files can be configured in the **General Parameters** form on the menu *Settings* → *General*. As default the file menu starts at the last directory selected.

For the formats of the files you are referred to Appendix A.

5.1.1 New project

Upon selecting *File* → *New Project*, all objects (land boundaries, polygons, splines, grids, samples, etc.) will be deleted; i.e. you start from scratch.

5.1.2 Open project

Upon selecting *File* → *Open Project,* the **Open Project** window appears in which you can browse to an existing project (<∗.d3d> file).

Remark:

- ◇ A project saved by QUICKIN or D-Waq DIDO can be read by RGFGRID.

5.1.3 Save project

Upon selecting *File* → *Save Project*, the current project (grid filenames and, if applicable, DD boundaries filename) will be saved under the same name. If the project name is not known yet, the **Save Project** window appears.

Remark:

- ◇ When you started with an existing project, or when you saved the project before, saving the project will not save changes you have made to the grid(s). Either use **Save Project As** or save individual grids.

5.1.4 Save project as

Upon selecting *File* → *Save Project As*, the current project can be saved under a different name.

5.1.5 Attribute files

On the *File* → *Attribute Files* sub-menu, see Figure 5.3, options are available to open and save objects that are indirectly related to the grids.

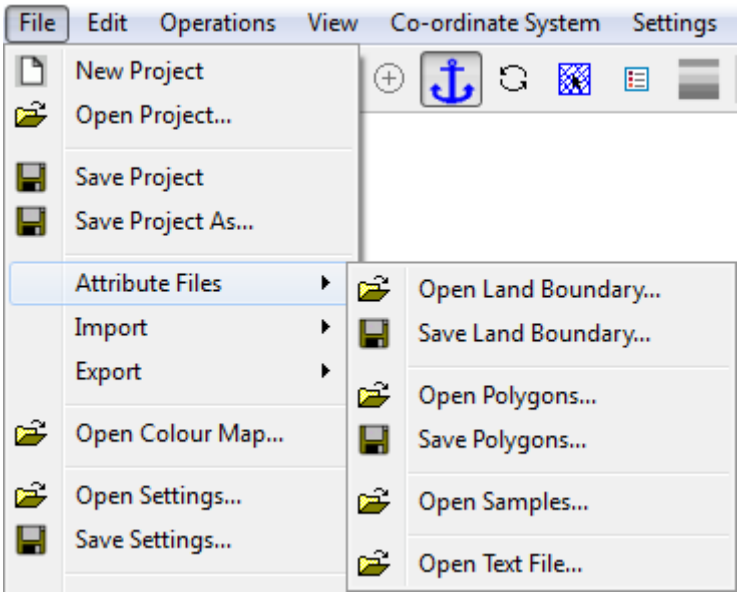


Figure 5.3: *Options on the* File→ Attribute Files *menu*

Open land boundary

Upon selecting *File* → *Attribute Files* → *Open Land Boundary*, you can open a collection of land boundaries. Land boundaries (or land-water marking) are in files with default mask <∗.ldb>. For a real application the land boundary is a guidance to define a grid for the model area.

Remark:

- ◇ If you open another land boundary file, it will be visualised together with the existing land boundary.

Open polygon

Upon selecting *File* → *Attribute Files* → *Polygons,* you can open a collection of polygons in a file with mask (<∗.pol>). Polygons are per definition closed. If the polygon is not closed in the file it will still be shown as closed.

Remark:

- ◇ If you open another polygons file, they will be visualised together with existing polygons.

Save polygons

When saving polygons, each polygon will be saved as a closed polyline. A polygon file has as default mask <∗.pol>.

Open samples

The bathymetry can be used as a guideline to determine the orientation and resolution of the required grid. This can be done visually, but also the grid design can take into account the samples. See *Settings* → *Orthogonalisation*, item *Design Method*, see Section 5.6.3.

The samples in a file with mask <∗.xyz>, may be a set of disordered x, y, z values given in a sequential list of free-formatted x, y, z values.

Remark:

- ◇ If you open another samples file, the samples will be visualised together with existing samples.

Open text file

Texts can be displayed in the graphics area if their position (x, y), the text and colour are defined. See an example in Appendix A.8.

5.1.6 Import

On the *Import* sub-menu, see Figure 5.4, options are available to import objects that are directly related to the grids.

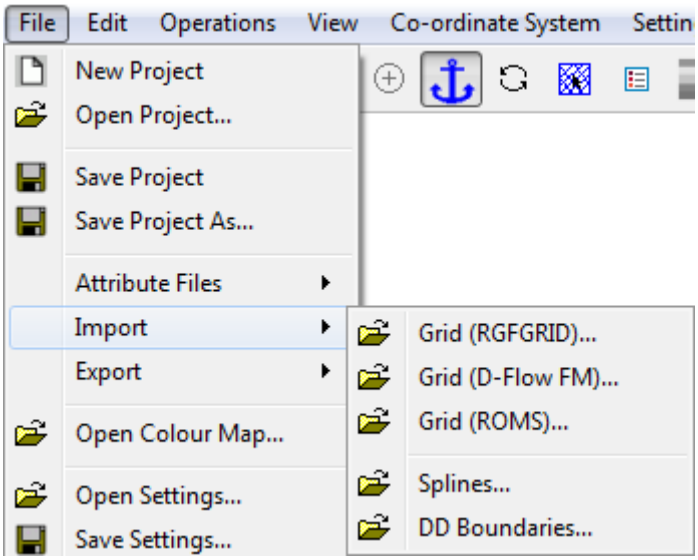


Figure 5.4: File → Import *menu options*

Grid (RGFGRID)

Upon selecting *File* → *Import* → *Grid (RGFGRID)*, you can open a collection of grids. The grid file has a default mask <∗.grd> or <∗_rgf.nc>.

Remarks:

- ◇ The co-ordinate system in RGFGRID is set accordingly to the system specified in the grid file.
- ◇ If the co-ordinate system is spherical then the co-ordinates are shown in stereographic projection.
- ◇ If no co-ordinate system is specified, Cartesian is presumed.

Grid (D-Flow FM)

Upon selecting *File* → *Import* → *Grid (D-Flow FM)*, you can open a collection of grids. The grid file has a default mask <∗_net.nc>.

Grid (ROMS)

Upon selecting *File* → *Import* → *Grid (ROMS)*, you can open a collection of regular grids in the NetCDF format off the Regional Ocean Modeling System. The grid file has a default mask <∗_roms.nc>.

Splines

The initial sketch of the grid is done by drawing splines. Splines are in files with default mask <∗.spl>.

Remark:

- ◇ If you open another splines file, the new splines will replace existing splines.

DD boundaries

In case of a domain decomposition application you will have multiple grids. How the grids are linked to each other is contained in the domain decomposition boundary file (ddb-file). The ddb-file will be made if you select *Operations* → *Compile DD Boundaries*, see Section 5.3.16.

5.1.7 Export

On the *File* → *Export* sub-menu, see Figure 5.5, options are available to export objects that are directly related to the grids.

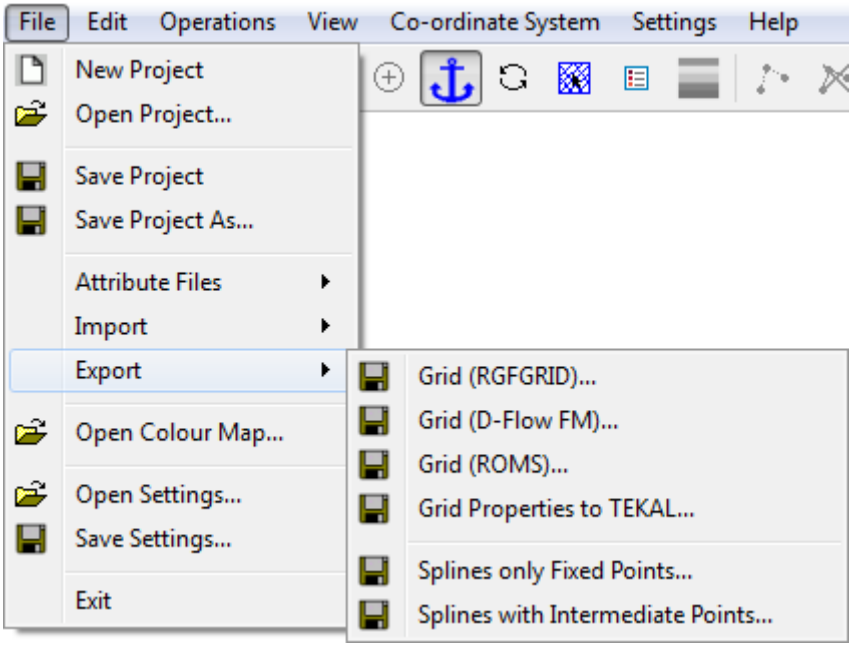


Figure 5.5: File → Export *sub-menu options*

Grid (RGFGRID)

The grid is saved in a file with mask <∗.grd> or <∗_rgf.nc>. Along with the <∗.grd> file, a second file is saved with mask <∗.enc>, containing the so-called grid enclosure, that outlines all active computational grid cells in Delft3D-FLOW.

Grid (D-Flow FM)

The grid is saved in the NetCDF file format suitable for D-Flow FM, the default mask <∗_net.nc> is used.

Grid (ROMS)

The grid is saved in the NetCDF file format suitable for the Regional Ocean Modeling System, the default mask <∗_roms.nc> is used.

Grid properties to TEKAL

The grid properties can be saved in a so-called TEKAL format, so that the properties can be visualised with Delft3D-QUICKPLOT or GPP, see QUICKPLOT (2013) and GPP (2013). The data is saved in a file with mask <∗.tek>, and contains the x, y co-ordinates, the orthogonality, the resolution, the smoothness, the curvatures, the grid sizes and the aspect ratios in columns.

Splines only fixed points

Splines are saved in a file with default mask <∗.spl>. Only those points which are visualised with a dot are stored in the file.

Splines with intermediate points

The splines including the intermediate points between the points visualised with a dot, can be saved in a file with default file mask <∗.spt>.

5.1.8 Open Colour map

You can choose from a number of pre-defined colour schemes (in file with masks <∗.clr> or <∗.clrmap>). These colour schemes have the same format as used for Delft3D-QUICKPLOT, see Appendix A.10 for the file format.

Restriction:

- ◇ Only the colour space `RGB` is supported

Remark:

- ◇ If the file <rgfgrid.clrmap> exists on the start-up directory then this file will be read, if the file does not exist on the start-up directory it will try to read the file on the installation directory <$D3D_HOME/$ARCH/plugins/default>.

5.1.9 Open Settings

If you have saved your RGFGRID settings in a previous session, you can open these settings again, see Appendix A.11 for the file format.

Remark:

- ◇ If the file <rgfgrid.ini> exists on the start-up directory then this file will be read, if the file does not exist on the start-up directory it try to read the file on the installation directory <$D3D_HOME/$ARCH/plugins/default>.

5.1.10 Save Settings

If you have made changes in one of the forms on the *Settings* menu, you can save these settings to be used later on again.

5.1.11 Exit

Exit from the RGFGRID program.

5.2 Edit menu

On the *Edit* menu, see Figure 5.6, several edit modes can be selected.

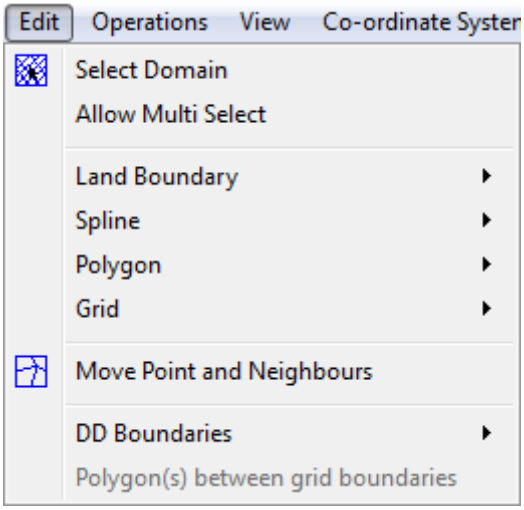


Figure 5.6: *Options on the* Edit *menu*

An edit mode is an operation mode which needs at least a mouse click, i.e. a set of operation instructions which is valid for a certain data set, and which may go with some specific display method. The following objects may be modified:

- ◇ Polygon
- ◇ Spline
- ◇ Grid
- ◇ DD Boundaries

`Esc` = Undo
In most edit modes, `Esc` will undo the latest action.

5.2.1 Select domain

If your project consists of multiple grids (so-called domain decomposition application) you can switch between the domains (grids) by clicking *Edit* → *Select Domain*, or click on the toolbar. Next, click on the grid you want to become the active grid.

5.2.2 Delete domain

To delete a domain or grid, select *Edit* → *Delete Domain*. Next, click on an active or inactive grid. When deleting a domain, also objects (except dredge and dump areas) defined on the grid will be deleted.

5.2.3 Allow Multi Select

When selecting option *Edit* → *Allow Multi Select* you are able to select more than one polyline of the land boundary, polygon or grid. For example, to merge several irregular grids use this option to select which domains need to be merged.

5.2.4 Land Boundary

The land boundary is used to visualise the land-water interface. To edit (define or modify) a land boundary, for possible edit actions see Figure 5.7.

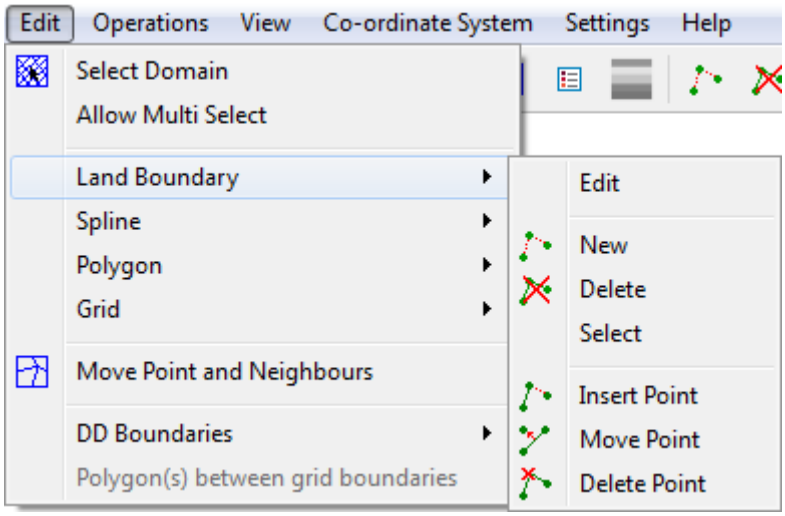


Figure 5.7: *Options on the* Edit → Land Boundary *menu*

5.2.4.1 Menu options

The key stroke to reach the menu item *Edit* → Land Boundary → *Edit* is: `CTRL+ALT+L`

Edit

Upon selecting *Edit* → *Land Boundary* → *Edit*, you can start editting a polyline that defines an Land Boundary. When there is no polyline the edit mode is set to *New*, otherwise you have to select first a polyline (from the menu *Edit* → *Land Boundary* → *Select* or press the key `s`). After you have selected the polyline you can use key-strokes, icons in the toolbar or menu items to switch the edit mode.

New

Upon selecting *Edit* → Land Boundary → *New*, you can start to define a new polyline, click on , or use the key-stroke `n` to start a new polyline.

Delete

Upon selecting *Edit* → *Land Boundary* → *Delete*, click on , or use the key-stroke `e`, to delete (erase) the selected polyline.

Select

Upon selecting *Edit* → *Land Boundary* → *Select*, or use the key-stroke `s`, you can select a polyline by clicking on one of its edges or vertices. After that the polyline will be highlighted

Insert point

Upon selecting *Edit* → *Land Boundary* → *Insert Point*, click on  , or use the key-stroke `i`, you can insert a point into the selected polyline. The point will be inserted at the nearest linear piece of the polyline.

Move point

Upon selecting *Edit* → *Land Boundary* → *Move Point*, click on  , or use the key-stroke `r`, you can move (replace) a point on the selected polyline.

Delete point

Upon selecting *Edit* → *Land Boundary* → *Delete Point*, click on  , or use the key-stroke `d`, you can delete a point on the selected polyline by indicating it.

5.2.4.2 Valid action keys are

The key stroke to reach the menu item *Edit* → Land Boundary → *Edit* is: `CTRL+ALT+L`

In *Edit* → *Land Boundary* mode the following keys can be used (mode is indicated in the statusbar):

- Key `a`: Add point after
  Add a point after the last point of a selected land boundary (after)
- Key `b`: Add point before
  Add a point before the first point of a selected land boundary (before)
- Key `d`: Delete single point
  delete single point in selected land boundary
- Key `i`: Insert single point
  Insert single point in the selected land boundary
- Key `r`: Move (replace) single point
  Replace single point in the select land boundary
- Key `s`: Select land boundary
  Select a spline
- Key `x`: Delete land boundary
  Delete a complete land boundary

5.2.5 Spline

On the *Edit* menu, point to *Spline*, see Figure 5.9

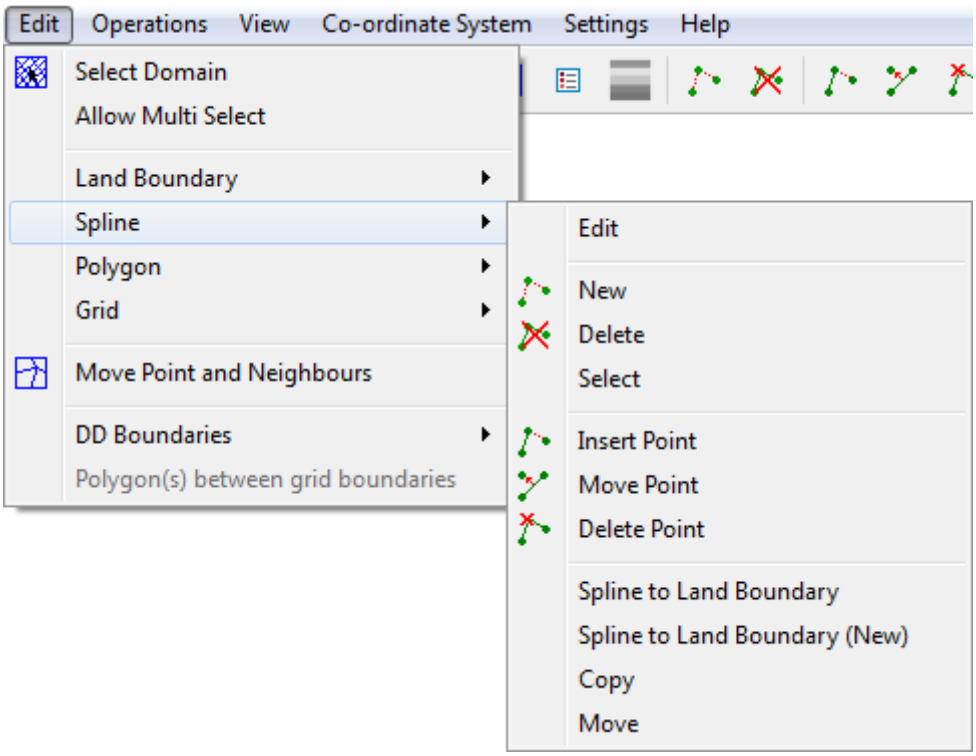


Figure 5.8: *Options on the* Edit → Spline *menu*

5.2.5.1 Menu options

The key stroke to reach the menu item *Edit* → Spline → *Edit* is: `CTRL+ALT+S`

Edit

Upon selecting *Edit* → *Spline* → *Edit*, you can start editting a spline (handled as a polyline). When there is no spline the edit mode is set to *New*, otherwise you have to select first a spline (from the menu *Edit* → *Spline* → *Select* or press the key `s`). After you have selected the spline you can use key-strokes, icons in the toolbar or menu items to switch the edit mode.

New

Up on selecting *Edit* → *Spline* → *New* you can start defining a new spline. Click the left mouse button at different positions to create a spline. To start a new spline, click the right mouse button and click the left mouse button again to create the next spline.

Delete

Up on selecting *Edit* → *Spline* → *Delete* you can delete a spline . Click with the left mouse button on a spline point, than that spline will be deleted.

Insert Point

Upon selecting *Edit* → *Spline* → *Insert Point*, you can insert a point. But first you have to select the spline in which you want to insert a point.

Move Point

Upon selecting *Edit* → *Spline* → *Move Point*, you can move a point on a spline. But first you have to select the spline in which you want to insert a point.

Delete Point

Upon selecting *Edit* → *Spline* → *Delete Point*, you can delete a point of a spline. But first you have to select the spline in which you want to insert a point.

5.2.5.2 Valid action keys are

The key stroke to reach the menu item *Edit* → Spline → *Edit* is: `CTRL+ALT+L`

In *Edit* → *Spline* mode the following keys can be used (mode is indicated in the statusbar):

- ◇ Key `a`: Add point after
  Add a point after the last point of a selected spline (after)
- ◇ Key `b`: Add point before
  Add a point before the first point of a selected spline (before)
- ◇ Key `d`: Delete single point
  delete single point in selected spline
- ◇ Key `i`: Insert single point
  Insert single point in the selected spline
- ◇ Key `r`: Move (replace) single point
  Replace single point in the select spline
- ◇ Key `s`: Select spline
  Select a spline
- ◇ Key `x`: Delete spline
  Delete a complete spline

5.2.6 Polygon

The polygon is used to limit the area of influence of operations and or edit actions. All grid points and samples that are inside the polygon are active in the subsequent interpolation or manipulation steps. The polygon is self closing.

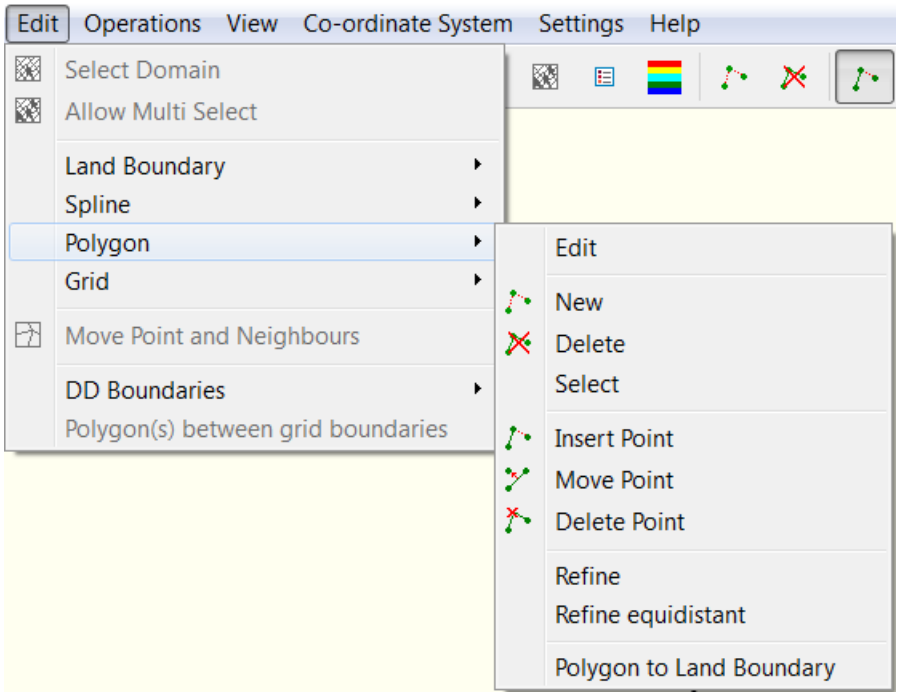


Figure 5.9: *Options on the* Edit → Polygon *menu*

5.2.6.1 Menu Options

The key stroke to reach the menu item *Edit* → Polygon → *Edit* is: `CTRL+ALT+P`

Edit

Upon selecting *Edit* → *Polygon* → *Edit*, you can start editting a polygon that defines an area of interest. When there is no polygon the edit mode is set to *New*, otherwise you have to select first a polygon (from the menu *Edit* → *Polygon* → *Select* or press the key `s`). After you have selected the polygon you can use key-strokes, icons in the toolbar or menu items to switch the edit mode.

New

Upon selecting *Edit* → Polygon → *New*, you can start to define a new polygon, click on , or use the key-stroke `n` to start a new polygon.

Delete

Upon selecting *Edit* → *Polygon* → *Delete*, click on , or use the key-stroke `e`, to delete (erase) the selected polygon.

Select

Upon selecting *Edit* → *Polygon* → *Select*, or use the key-stroke `s`, you can select a polygon by clicking on one of its edges or vertices. AFter that the polygon will be highlighted

Insert point

Upon selecting *Edit* → *Polygon* → *Insert Point*, click on  , or use the key-stroke `i`, you can insert a point into the selected polygon. The point will be inserted at the nearest linear piece of the polygon.

Move point

Upon selecting *Edit* → *Polygon* → *Move Point*, click on  , or use the key-stroke `r`, you can move (replace) a point on the selected polygon.

Delete point

Upon selecting *Edit* → *Polygon* → *Delete Point*, click on  , or use the key-stroke `d`, you can delete a point on the selected polygon by indicating it.

Refine

Choosing the option *Edit* → *Polygon* → *Refine* you are able to refine the selected polygon. This is done by selecting two vertices of the polygon. When selecting the two vertices the part where the refinement will be take place is marked with circles instead of dots. If you need the other part of the polygon hit the `i` to invert the selection.

The algorithm for the refinement is as follows:
The distance between the first selected point and marked neighbour, and the distance between the last selected point and its marked neighbour is determined. The refinement distance will be the linear interpolation between this two values, so a gradual change of distance along the marked part of the polygon. Exiting vertices are replace by the new ones.

Refine equidistant

To insert points to obtain an equidistant refinement on the edges of a polygon, select *Edit* → *Polygon* → *Refine equidistant*. Next, click on one or more edges from the polygon to select or deselect them, or double-click to select or deselect all edges from the polygon.

Keys actions

In *Edit* → *Polygon* → *Refine equidistant* mode the following keys can be used:

- ◇ Key `p` = Polyline refine
  Pressing `p` allows you to refine the selected polyline(s) of the polygon using a distribution ds. The message at the left of the statusbar now reads 'Press Return to accept Refinement or P to change Refinement'.
  Pressing `Return` will refine the polyline(s) within the polygon using the current value of ds, pressing `p` will allow the user to redefine ds.
- ◇ Key `l` = Snap selected polyline(s) to landboundary
  Pressing `l` allows you to snap selected polyline(s) to an existing landboundary. The message at the left of the statusbar now reads 'Press Return to accept attachment or S to (re)select segments'.

Pressing `Return` will snap the selected polyline(s) to the landboundary, pressing `s` will allow the user to (re)select polyline(s) within the polygon.

Polygon to Land Boundary

Through the *Edit* → *Polygon* → *Polygon to Land Boundary* option, a subset of a polygon's vertices can be selected and snapped to the nearest land boundary. In case no polygon is currently selected, the user is asked to select one by clicking near a polygon vertex. With a polygon selected, the first vertex can be selected by clicking near it. Clicking the same vertex for a second time deselects it. By selecting a second vertex, a subset of vertex points is snapped to the nearest land boundary. The subset consists of vertices [n,m] if m $>$ n or all but [m+1,n-1] if m $<$ n.

5.2.6.2 Valid action keys are

The key stroke to reach the menu item *Edit* → Polygon → *Edit* is: `CTRL+ALT+P`

In *Edit* → *Polygon* mode the following keys can be used:

- ◇ Key `I`: Insert
  In *Edit* → *Polygon*, pressing `I` starts the vertex insert action depending on the first click on the screen, there are two options.

Normal	If the first click is in between two vertices of the polygon then a point will be inserted in the closest edge.
Incremental	If the first click is on a vertex of the existing polygon the the incremental insert option is started, which means all next locations click by the left mouse button are added to the polygon behind the indicated point. This mode will be cancelled when indication a existing point of the polygon and the normal insert option will be used. Hitting another key will also cancel this insert option.

  The message at the left of the statusbar now reads 'Insert a point', click the left mouse to insert individual points.
- ◇ Key `r`: Replace
  Pressing `r` allows you to replace (move) individual polygon points. The message at the left of the statusbar now reads 'Replace: Get a Vertex'. If you have got it by clicking the left mouse, the message will read 'Replace: Put a Vertex', and you can do so by clicking the left mouse at the new desired position.
- ◇ Key `d`: Delete
  Pressing `d` allows you to delete individual polygon points.
- ◇ Key `x`: Break open polygon
  Keep *X* pressed and move with the cursor over a polygon point to split the polygon. Effectively, the pointed co-ordinates are replaced by default 'missing' values.
- ◇ Key `e` : Erase polygon
  Entire polygon sections are deleted. Press key `e` and then click with the left mouse button on a point of the polygon which need to be deleted. Finish the operation by pressing the right mouse button.

5.2.7 Grid

On the *Edit* menu, point to *Grid* you can edit the grid, see Figure 5.10

The key stroke to reach the menu item *Edit* → Grid → *Edit* is: `CTRL+ALT+G`

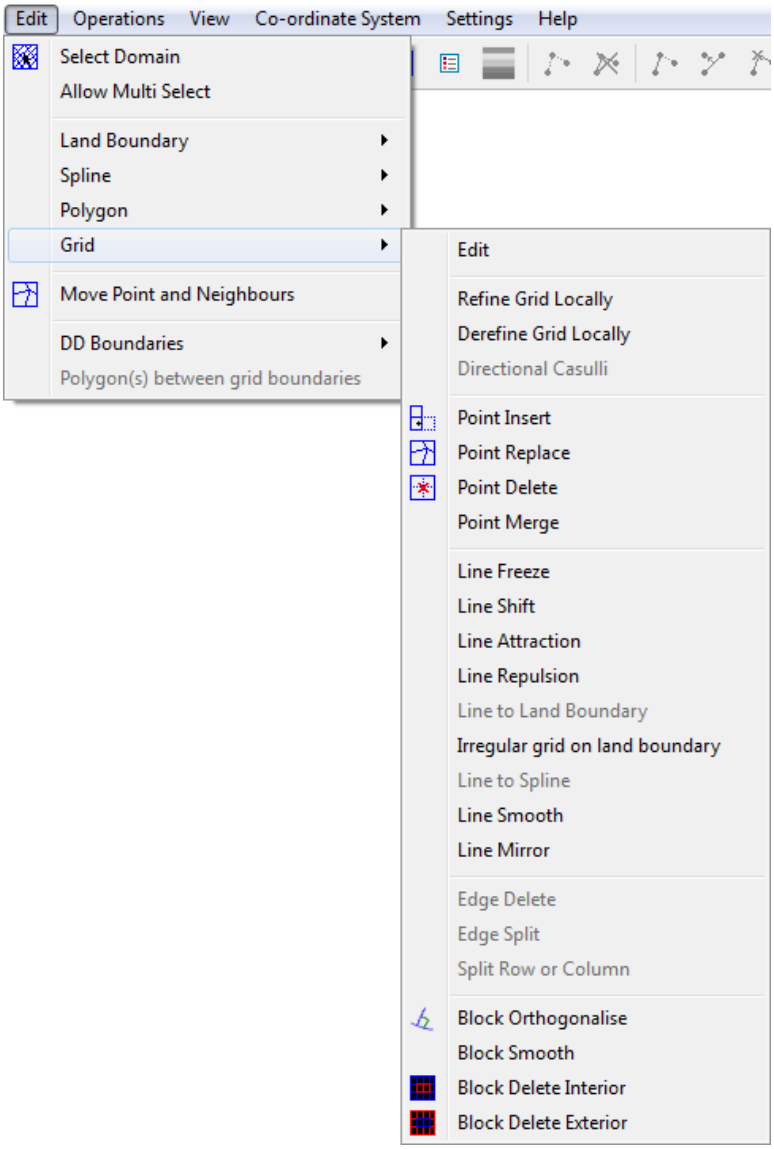


Figure 5.10: *Options on the* Edit → Grid *menu*

5.2.7.1 Menu Options

Edit

Upon selecting *Edit* → *Grid* → *Edit*, you can start editing a grid. When there is no grid the edit mode is set to *New*, which menas start editing an irregular grid. Otherwise you have to select first a grid (from the menu *Edit* → *Grid* → *Select* or press the key `s`). After you have selected the grid you can use key-strokes, icons in the toolbar or menu items to switch the edit mode.

Refine grid locally

This option operates on part of the grid and the direction depends on the grid line indicated by you.

First you specify (on the *Settings* → *General* menu) the number of times that the grid has to be refined in the M- or N-direction (see Section 5.3.10). Then you indicate 2 points on a grid line between which the refinement has to be performed.

Derefine grid locally

This option operates on part of the grid and the direction depends on the grid line indicated by you. This operation is the opposite of *Refine Grid Locally*. First you specify (menu *Settings* → *General)*, the number of times that the grid has to be de-refined in the M or N direction (see Section 5.3.10). Then you indicate on a grid line 2 points between which the de-refinement has to be performed. Next, smooth the jump in grid sizes.

Directional Casulli

TODO(??): Under construction

5.2.7.2 Point

On the *Edit* menu, point to *Grid* and click on one of the options to operate on individual grid points. To insert, delete or move grid points you can either use the menu options, the icons on the toolbar, or the keyboard to switch between these operations.

After selecting one of the options: insert, move or delete point the program is in point edit mode

Point Insert

Press the `I`-key, use the toolbar icon  or click the menu item *Edit* → *Point* → *Insert* to bring the progrma into insert mode.

If the program is in insert mode, (message 'Select Grid Cell' at the lower left side of the screen), click the left mouse inside a grid cell to create a new grid cell at the border of the grid. The indicated grid cell will be 'mirrored' to the grid cell side closest to the clicking point.

Point Replace

Press the `R`-key (Replace), use the toolbar icon  or click the menu item *Edit* → *Point* → *Move* to bring the program into replace mode.

The message at the lower left of the screen now reads 'Get a point'. Click left to indicate a grid point; the message will read 'Put a point'. Move the cursor to the desired position and click left again.

Point Delete

Press the D-key, use the toolbar icon  or click the menu item *Edit* → *Point* → *Delete* to bring the program into delete mode.

If the program is in delete mode, delete grid points by just clicking them.

Points Merge

Irregular grid: Upon selecting *Edit* → *Grid* → *Points Merge* two nodes can be merged. Select a node by the left mouse button and than select a node to which the first selected node is merged to.

5.2.7.3 Line

The operations line freeze, line shift, line attraction, line repulsion and line smooth operate in line mode, see Figure 5.10.

They all use the same procedure to indicate a line and an influence area.

You first indicate a line by marking its end points, using the left mouse; next you indicate the influence area by marking one or two grid-points at one or both side of the line, respectively. Pressing Esc enables the replacement of the last added point; pressing Esc+Esc cancels all the selected block points, after you may redo the selection procedure. You click the right mouse for the final selection of line and area. After the indication you perform the operation (e.g. line shifting, attraction or repulsion). The result can still be reversed (by pressing several times the Esc key).

Line Freeze

Frozen lines are grid lines that are kept fixed in the orthogonalisation process. That is, the end points are kept fixed and the points in between can only move in the direction along the grid line. Frozen lines can be edited by clicking 2 points that lie on the same grid line. You can unfreeze grid lines by first pressing the D key and click with the left mouse on one of the endpoints. You can also use I (insert) mode to define lines to freeze.

Line Shift

This option provides the possibility to fit the grid's edges to a land boundary. First you indicate a line and indicate the influence area. Then, you can shift the line by shifting some or all of the individual points of that line. The end points can also be shifted. After clicking the right mouse to indicate that the line has been put into the correct new position, the points on the line between the end-points will be shifted by linear interpolation between all repositioned points. Then, a field transformation will be performed in the influence area, with centre points that are now consecutive points on the shifted centre line. If you are not satisfied with the transformed result, press several times the Esc key. You will then be put back into *Edit* → *Line* → *Shift* mode. You can carry on shifting lines by simply repeating the same sequence of actions.

Line Attraction

Here, you have again to 'Indicate a line', by marking its end points, and to 'Indicate an influence area' (see *Edit* → *Line* → *Shift*). The grid will be attracted to the indicated line, making use of the line transformation described above, in the field indicated by the influence area.

In *Settings* → *General* the parameter *Attraction/Repulsion Parameter* can be changed, see Figure 5.42.

Line Repulsion

The reverse of *Edit* → *Line* → *Attraction*.

In *Settings* → *General* the parameter *Attraction/Repulsion Parameter* can be changed, see Figure 5.42.

Line to Land Boundary

The edge of the grid can be fitted to a land boundary by hand, using the *Edit* → *Line* → *Shift* option, or automatically, using the present option, *Edit* → *Line* → *Line to Land Boundary*. The automatic option may not always deliver exactly what you want. This can be caused by irregular shapes in the land boundary. However, we do not want to be compelled to analyse and polish up the land boundary a priori, in the digitising phase.

Therefore, both the automated and hand option are included in the program. Just indicate the first and last point of the line that you want to fit to the land boundary. Then click the right mouse. Next, all intermediate points will be translated to their nearest land boundary. Then, a line shift will be performed, equal to the one mentioned above, shifting the indicated line and the surrounding grid. Press `Esc` three times if the result is unsatisfactory. The original grid will then be restored. The algorithm which decides to which land boundary line segment the grid line should be attracted, first looks for the closest land boundary point. An error may occur here, if the closest land boundary line segment is very long, and land boundary points of other segments are more close to the indicated grid line. In that case open the land boundary as a polygon and add (insert) some points to the long land boundary segment, so that points on this segment are closest to the indicated grid line.

Irregular grid on Land Boundary

TODO(??): Under construction

Line to spline

Similar as line to land boundary. If you do not need the spline grid anymore, first delete the splines and then draw just 1 spline to which you want to attach the grid.

Line Smooth

You have to 'Indicate a line', by marking its end points, and to 'Indicate an influence area' (see *Edit* → *Line* → *Shift*). Within this area, the grid will be smoothed into the direction indicated by the line.

The smoothing process can be configured, see Section 5.6.1, parameters *Number Smoothing Iterations* and *Smoothing Parameter*.

Line Mirror

Indicate a grid line at the edge of the grid by marking its end points. Click right to execute the mirror process; grid cells will be created. After this the operation can be repeated by using the key `CTRL+M`

5.2.7.4 Block

Block delete, block cut, block orthogonalise and block smooth all operate in block mode, see Figure 5.10. An influence area (block) is indicated by clicking two, three or four points.

Block orthogonalise

Click two, three, or four points to indicate the corners of the grid block. A minimal block is selected which just contains the selected points. Press `Esc` if you want to replace the latest indicated point, press `Esc+Esc` to redo the selection of the block. Clicking right results in the orthogonalisation of the grid inside the selected block. Press `Esc+Esc+Esc` if you want to cancel the latest action, or click *Undo* on the *Operations* menu.

You can specify parameters that control the orthogonalisation in *Settings* → *Orthogonalisation*, see Figure 5.44.

Block smooth

Click two, three, or four points to indicate the corners of the grid block. A minimal block is selected which just contains the selected points. Press `Esc` if you want to replace the latest indicated point, press `Esc+Esc` to redo the selection of the block. Clicking the right mouse results in the smoothing of the grid inside the selected block. Press `Esc+Esc+Esc` if you want to cancel the latest action.

The smoothing process can be configured, see *Settings* → *General*, parameters *Number Smoothing Iterations* and *Smoothing Parameter*, see Figure 5.42.

Block delete interior

Click two points to indicate the corners of the grid block that you want to delete. A minimal block is selected which just contains the selected points. Clicking right results in the annihilation of the block area. Press `Esc` if you want to replace the latest indicated point, press `Esc+Esc` to redo the selection of the block. Press `Esc+Esc+Esc` if you want to cancel the latest action, or select *Undo* on the *Operations* menu.

Block delete exterior

Click two points to indicate the corners of the grid block. A minimal block is selected which just contains the selected points. Clicking right results in the annihilation of the grid in the area outside the selected block. Press `Esc` if you want to replace the latest indicated point, press `Esc+Esc` to redo the selection of the block. Press `Esc+Esc+Esc` if you want to cancel the latest action, or click *Undo* on the *Operations* menu.

5.2.8 Move Point and Neighbours

Modifications will be made by shifting the centre point of a field of points. The field transformation is based upon the relative shift of the centre point. For all cells in the vicinity of the centre, that shift is transformed to their local grid cell orientation and will be decreased in magnitude in proportion to the physical distance to the centre cell. In that way a quasi-orthogonal transformation is induced. The area of influence is always one sixth of the area that is currently displayed on the screen. (So, if you want to decrease or increase the area of influence, zoom in or zoom out).

5.2.9 DD Boundaries

This option is only relevant to users of the Delft3D domain-decomposition system, or if you want to keep some parts of the boundary fixed in the orthogonalisation.

At the interface between two grids of a multi-domain model (DD Boundaries) the grids should satisfy the following rules:

- ◇ At sub-domain interfaces the grids should be nicely connected (no overlap and “no holes” between sub-domains).
- ◇ In case of horizontal grid refinement, grid lines in the coarse domain should be continued in the fine sub-domain, see Figure 5.11. Thus, there should be a 1-to-N refinement, with N an integer number.

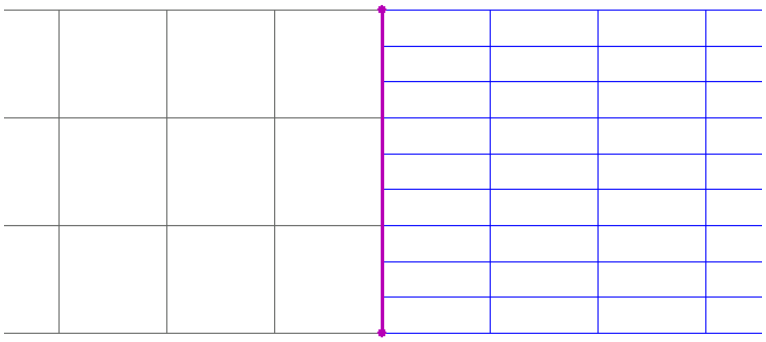


Figure 5.11: *Example of a 1-to-3 refinement along a DD boundary*

◇ Each grid line should cover or be covered by another grid line. The domain decomposition of Figure 5.12 does not fulfil this requirement. Although the DD-boudaries A and B have a correct refinement factor.

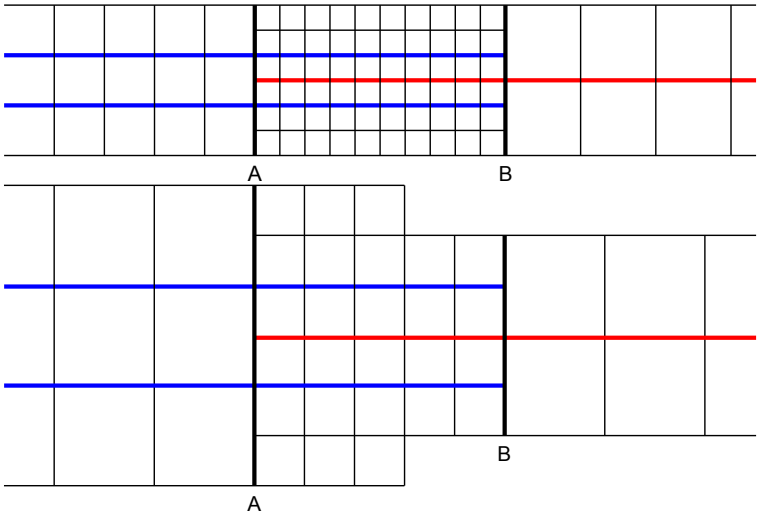


Figure 5.12: *Two examples of not allowed domain decompositions, although both DD-boundaries (A and B) satisfy the refinement condition; the red line and blue lines do not cover each other*

◇ Grids must be of the same type (thus, all in spherical co-ordinates, or all in Cartesian co-ordinates).
◇ The grid orientation should be the same (increasing M- and N-numbering in the same direction).
◇ No coupling of columns to rows or vice versa.
◇ Sub-domain interfaces should be straight lines (no stair-case interfaces).

DD boundaries can be edited by clicking boundary points that lie on the same grid line, see Figure 5.13. You can delete boundary points by first pressing the `D` key and click with the left mouse. `R` (replace) mode and `I` (insert) mode are also available. The specified boundaries are saved together with the grid in a file with mask $<*$.ddb$>$. This file is created when selecting *Operations* $\rightarrow$ *Compile DD Boundaries*. See Appendix A.9 for the format of this file.

DD boundaries are also used in the orthogonalisation process. Because DD boundaries can only be located on boundary points, their administration can be used to fix boundary points in the orthogonalisation process.

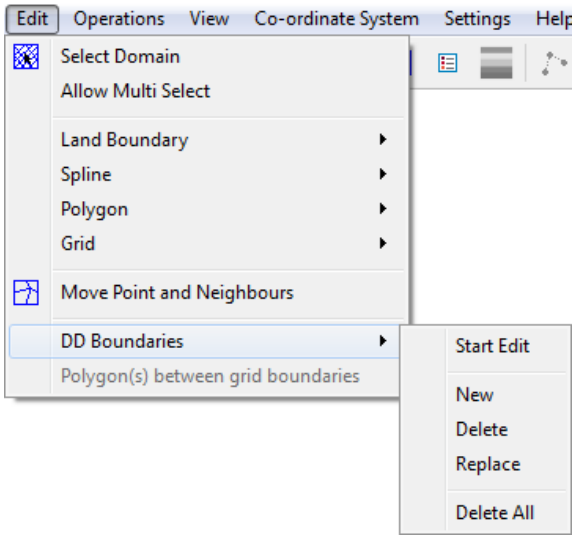


Figure 5.13: *Options on the* Edit → DD Boundaries *menu*

New

Start defining a new DD boundary.

Delete

Delete a single point of a DD boundary.

Replace

Replace a single point of a DD boundary.

Delete All

Delete all domain-decomposition boundaries.

Remark:

- ◇ DD Boundaries can also be defined in a single domain, see Figure 5.14.

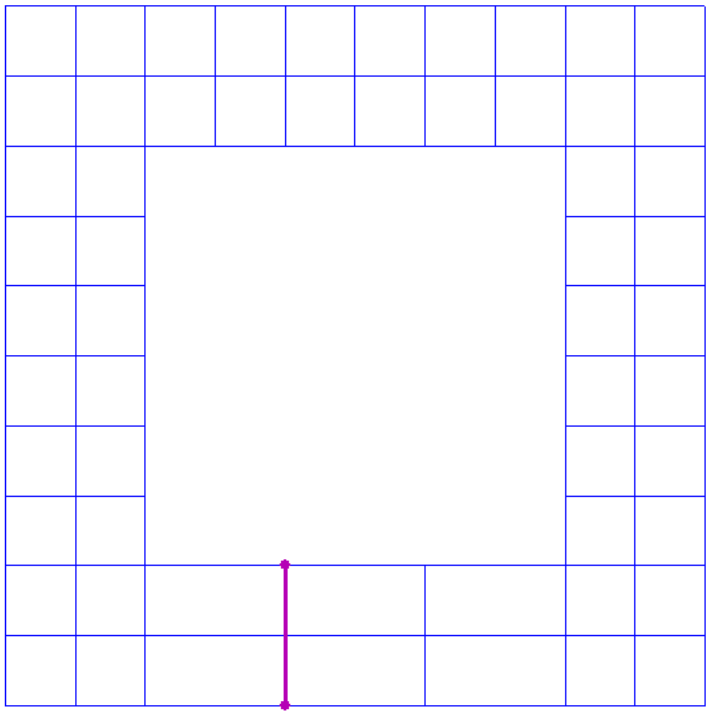


Figure 5.14: *DD Boundary in a single domain*

5.2.10 Polygon(s) between grid boundaries

The aim of this option is to define a polygon in which a irregular grid can be generated, this irregular grid can than be concatenate to the existing irregular grids.

In the *Edit* → *Polygon(s) between grid boundaries* mode a polygon can be defined which take care of the grid nodes at the existing and selected (computational) boundaries of the meshes. The polygon will consist of vertices which are attached to the boundary grid nodes of the selected grids. After wards this polygon can be handled with the normal polygon edit options to adjust the shape of the polygon. Be sure that you do not move the vertices which are locate at the grid boundaries.

The procedure is as follows: Select a point on the boundary from the first grid, select a second point aon the boundary of the same grid or another grid. If two points are selected on the boundary of the same grid then all boundary points in between these two points are selected, using a shortest node-path algorithm. To en the selection press the right mouse button or select the very first point again. Now this polygon editing behaviour will fall back to the normal polygon editing mode. If you are satisfied then generated the irregular grid by selecting *Operations* → *Create grid from polygon* from the menu bar. To merge the grids, select all grids which you want to merge and than select *Operations* → *Merge grids*. If you accidentally moved one vertex of the polygon which should lie on the grid boundary (and it was not merged) then you can merge these nodes by using the option *Edit* → *Grids* → *Merge node*

5.3 Operations menu

On the *Operations* menu, see Figure 5.16, you may choose to generate a grid from a set of splines and to perform various operations on the grid (create, de/refine, orthogonalise, compile dd, etc.). Refinement and orthogonalisation parameters must be changed on the *Settings* menu. Operations at individual grid points can be selected on the *Edit* menu.

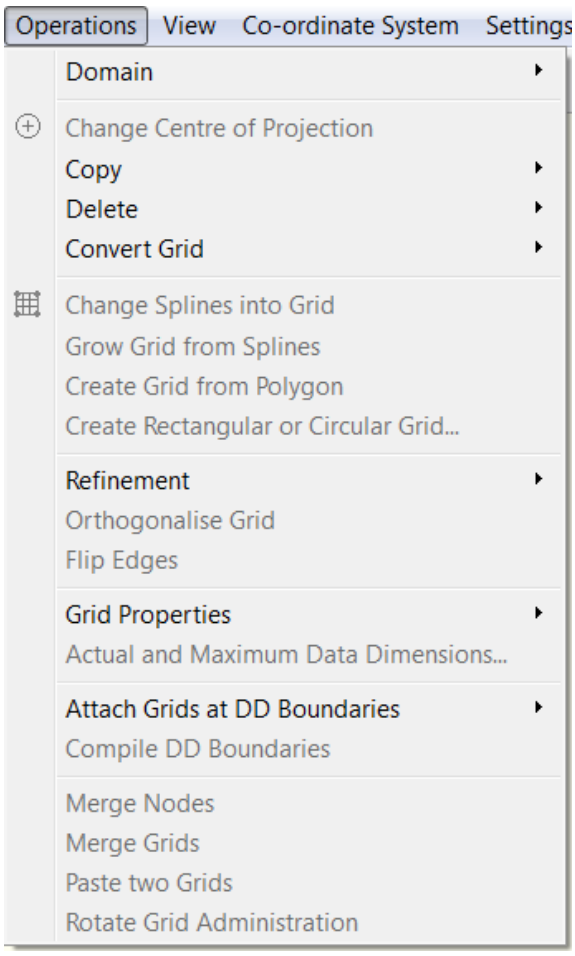


Figure 5.15: *Options on the* Operations *menu*

5.3.1 Domain

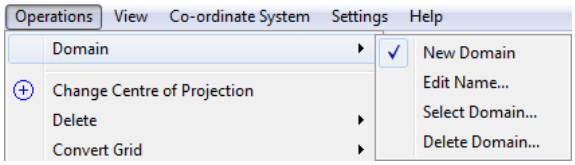


Figure 5.16: *Options on the* Operations *menu*

New domain

When selecting *Operations* → *Domain* → *New Domain*, a new domain is created. In addition, it enables the new domain mode, which means that every subsequent grid create action will add a new domain to your model. By default, the add points to irregular grid operation is active. This means that if one starts clicking points on the canvas, a new irregular grid will be

made and added to the current domain.

Edit name

When selecting *Operations* → *Domain* → *Edit Name...* you are able to change the name of the domain. This is mostly needed to change the default name of the domain.

Select domain

When selecting *Operations* → *Domain* → *Select Domain...* you are to select a domain from the list of domains. The menu option *Edit* → *Multi Allowed* select is should be ticked off.

Delete domain

When selecting *Operations* → *Domain* → *Delete Domain...* you are to delete a domain from the list of domains.

5.3.2 Change centre of projection

For spherical co-ordinates RGFGRID can use two different projections, plane projection and stereographic projection. For stereographic projection a special function is implemented to centre the computer screen to the centre of projection and the sphere. This function can be invoked by clicking the menu item *Operations* → *Change Centre of Projection* see Figure 5.16. When using this command the centre of the projection is set to the centre of the screen. This action requires recalculation of the projection and a new screen refresh. The centre of the projection does not change when using zoom in, zoom out or pan, so there is no performance drawback and a smooth screen-refresh is obtained.

5.3.3 Copy

The *Operations* → *Copy* → ... options, see Figure 5.17, operate on either all features, or only on the selected features. Relevant features are land boundaries, splines, polygons and grids.

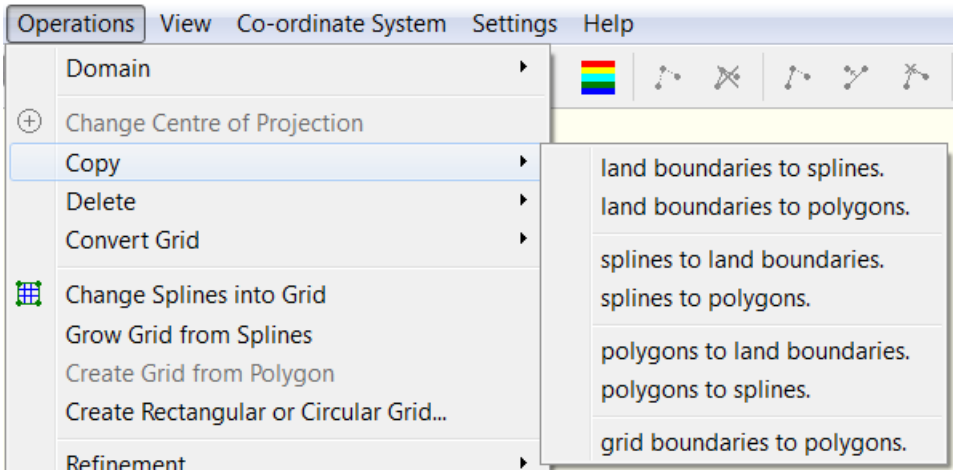


Figure 5.17: *Options on the* Operations → Copy *menu*

land boundaries to splines.

This operation copies land boundaries and converts those to splines.

land boundaries to polygons.

This operation copies land boundaries and converts those to polygons.

splines to land boundaries.

This operation copies splines and converts those to land boundaries.

splines to polygons.

This operation copies splines and converts those to polygons.

polygons to land boundaries.

This operation copies polygons and converts those to land boundaries.

polygons to splines.

This operation copies polygons and converts those to splines.

grid boundaries to polygons.

This operation copies grid boundaries and converts those to polygons.

5.3.4 Delete

On the *Operations* → *Delete* menu, see Figure 5.19, you may choose to delete land boundary, splines, grid, polygons, samples or elements.

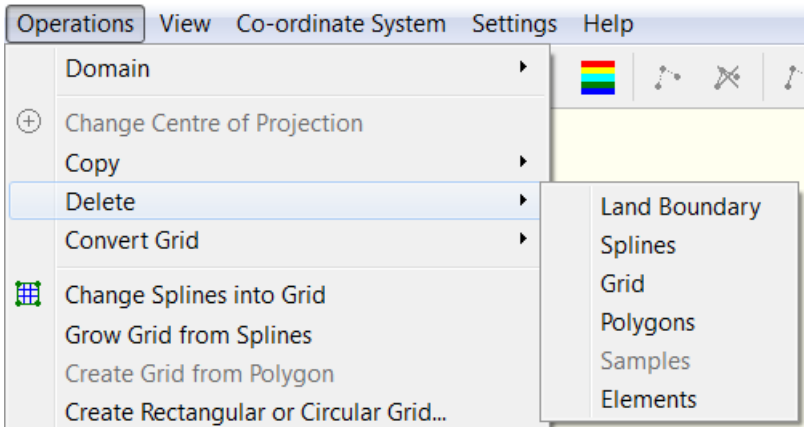


Figure 5.18: *Options on the* Operations → Delete *menu*

Land boundary

If a polygon is present the land boundary points inside the polygon will be deleted. If more polygons are defined only the first polygon will be used. If no polygon is defined you are asked if you want to delete all land boundary points.

Splines

Delete all the splines in the spline grid.

Grid

If a polygon is present the grid elements (partly) inside the polygon will be deleted. If more polygons are defined only the first polygon will be used. If no polygon is defined you are asked if you want to delete all grid points.

Polygons

Delete the polygon(s).

Samples

Delete all samples within a polygon. If more polygons are defined only the first polygon will be used. If no polygon is defined, you are asked if all samples must be deleted.

Elements

If a polygon is present, the grid elements with their centre of gravity within the polygon are deleted. Otherwise all grid elements are deleted, which is effectively the same as the before mentioned *Operations* → *Delete* → *Grid* option.

5.3.5 Convert grid

Upon selecting *Operations* → *Covert Grid* you are able to convert a regular to irregular grid or the other way around.

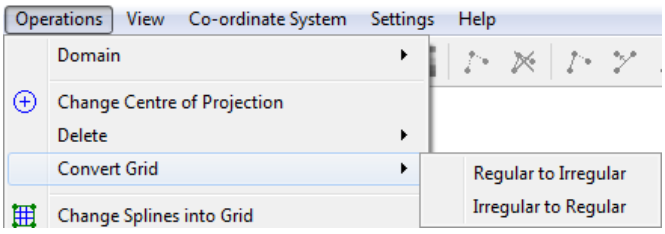


Figure 5.19: *Options on the* Operations → Convert Grid *menu*

Regular to Irregular

Convert the selected regular grid to an irregular grid.

Irregular to regular

Convert the selected irregular grid to a regular grid. Some times you need to select *Operations* → *Rotate Grid Administration* several times to get the orientation of the grid indices in the right orde.

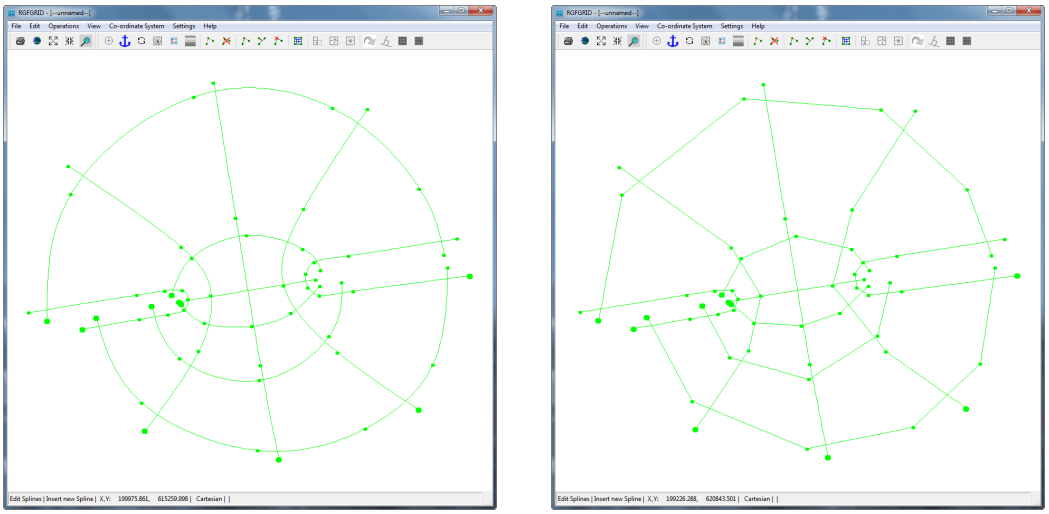
5.3.6 Change splines into grid

This operation can also be activated from the toolbar by clicking  .

The splines are ordered and directly refined into a regular grid. The refinement factors can be specified by selecting *Settings* → *General* and specifying the *M-Refinement Factor* and *N-Refinement Factor*, see Figure 5.42. Spline intersection points can only be identified if the straight lines between the control points of two splines intersect. To check this visually, you can display the splines as straight lines (see Figure 5.20, this can be set in the *Settings* → *General* form, parameter *Line or Spline Representation*. The correct ordering is only possible if a consistent result-grid is feasible.

At present, the spline-grid must satisfy the following restrictions:

- ◇ The set of splines need to be topological equivalent with a rectangle.
- ◇ Splines may not intersect twice or intersect themselves
- ◇ Splines with the same orientation may not intersect



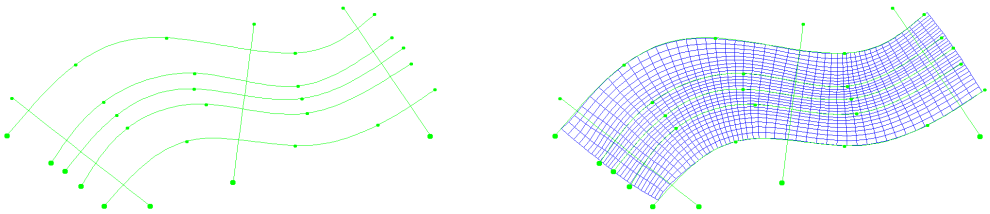
(a) *Curved representation* ***(b)*** *Linear representation*

Figure 5.20: *Different representation of splines*

The smoothness of the result-grid can be influenced by specifying the parameter *Equidistant or Smooth Interpolation* in *Settings* → *General,* see Figure 5.42.

5.3.7 Grow grid from splines

When selecting *operations* → *Grow Grid from Spline* a regular grid will be generated from a centre spline. This option is especially suitable to generate a grid for river simulations. The user is required to provide a spline, from which the grid is grown perpendicularly. Note that the grid can be grown from multiple centre splines simultaneously. Per centre spline, the extent of the grid and the heights of the grid layers can be controlled by supplementary splines and setting parameters (see menu option *Settings* → *Grow Grid from Splines...*). See Figure 5.46.

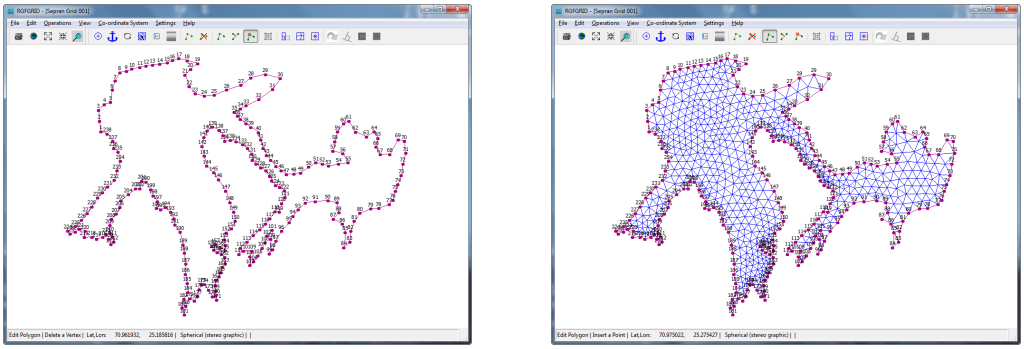


(a) *Splines* ***(b)*** *Generated grid from splines*

Figure 5.21: *Create grid from splines with option* Grow Grid from Spline

5.3.8 Create grid from polygon

When selecting *operations* → *Create Grid from Polygon* a irregular grid will be generated from the selected polygon (see Figure 5.22).

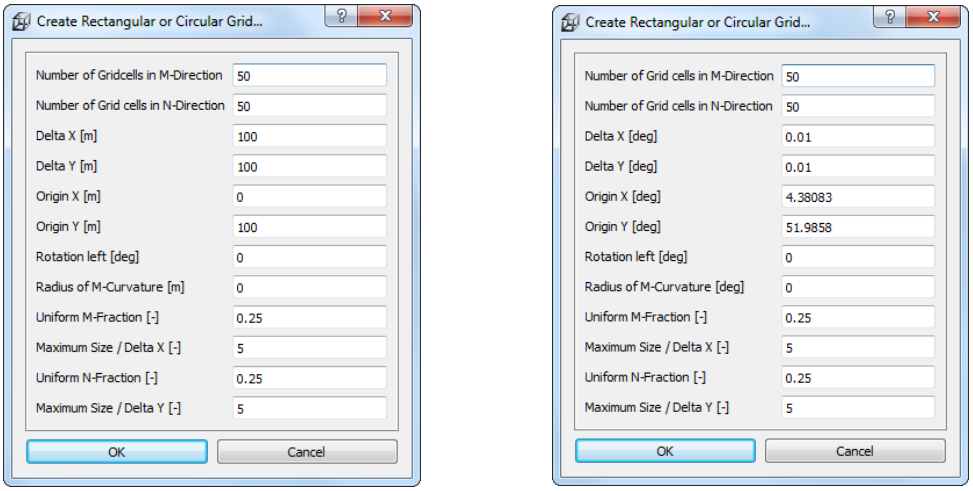


(a) *Selected polygon* ***(b)*** *Generated grid from polygon*

Figure 5.22: *Create grid from selected polygon*

5.3.9 Create rectangular or circular grid

Specify the grid spacing, grid origin and the number of grid cells in both directions to quickly create a rectangular grid. Grid sizes may be increased in size towards the boundaries by specifying the ration of the maximum grid-size at the boundaries relative to the size of the uniform fraction. The uniform fraction is the number of grid cells with uniform spacing vs. the total number of grid cells in a direction. A circular grid is created if the radius of curvature is non-zero. In that case, the grid origin is interpreted as its centre point. The parameters involved are, see Figure 5.23.



(a) *Cartesian grid* ***(b)*** *Spherical grid*

Figure 5.23: Parameters for Rectangular or Circular Grid *form.*

The default settings are:

- ◇ *Number of Grid Cells in M-Direction* default: 50
- ◇ *Number of Grid Cells in N-Direction* default: 50
- ◇ *Delta X [m] or [deg]* default: 100.0 or 0.01
  Grid cell size M-direction [m] or [deg]
- ◇ *Delta Y [m] or [deg]* default: 100.0 or 0.01
  Grid cell size N-direction [m] or [deg]
- ◇ *Origin X [m] or [deg]* default: 0.0 or 4.3803
- ◇ *Origin Y [m] or [deg]* default: 100.0 or 51.9858
- ◇ *Rotation left [deg]* default: 0.0
- ◇ *Radius of M-Curvature [m]* default: 0.0
- ◇ *Uniform M-Fraction [-]* default: 0.25
  Fraction of grid cells which contains the default grid size (ex. 0.25*50 = 13+1 grid cells with width size of 100 [m]))
- ◇ *Maximum Size / Delta X [-]* default: 5.0
- ◇ *Uniform N-Fraction [-]* default: 0.25
  Fraction of grid cells which contains the default grid size (ex. 0.25*50 = 13+1 grid cells with width size of 100 [m]))
- ◇ *Maximum Size / Delta Y [-]* default: 5.0

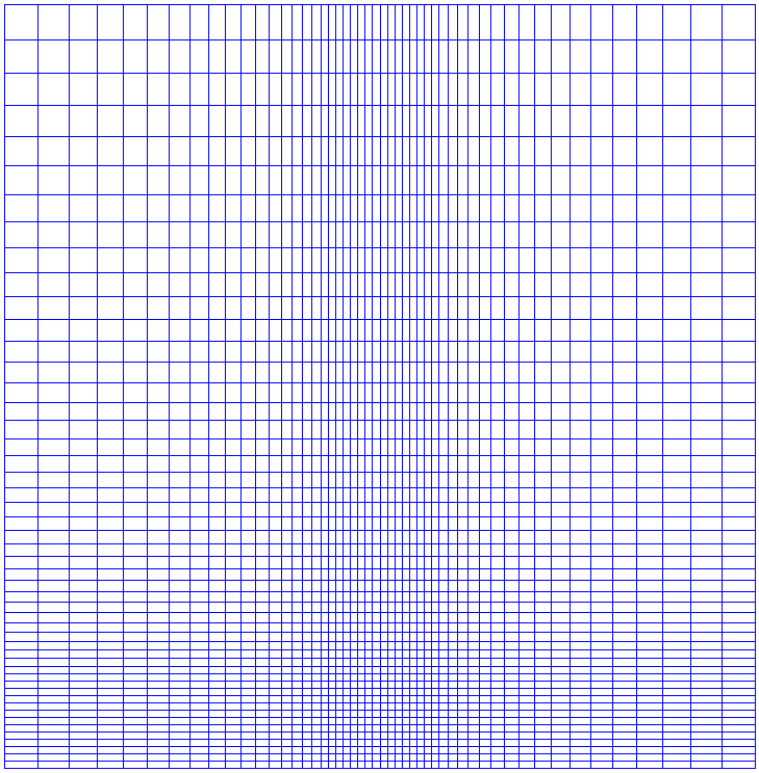


Figure 5.24: *Rectangular grid, created with* Maximum Size / Delta X *= "5" and* Maximum Size / Delta Y *= "5"*

5.3.10 Refinement

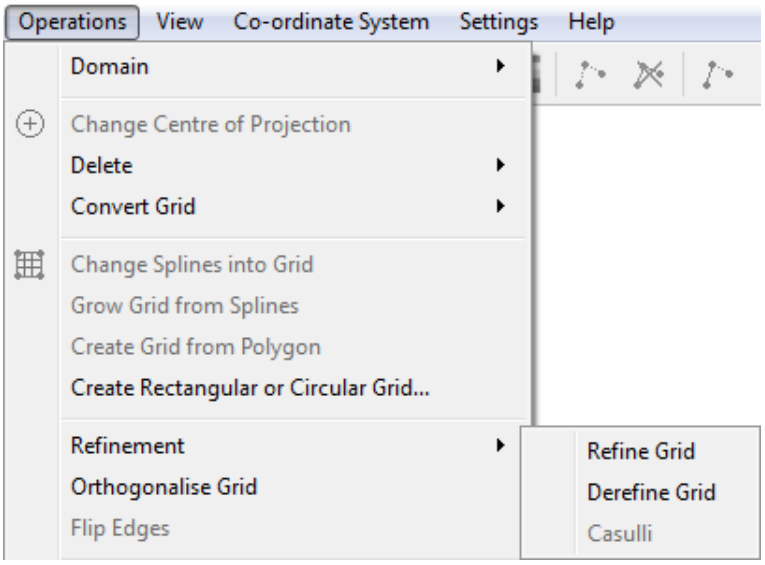


Figure 5.25: *Options on the* Operations → Refinement *menu*

Refine grid

This option operates on the whole grid and in both directions.

In *Settings* → *General* you first specify the number of times you want to refine the grid. The parameters for the M and N direction are *M-Refinement Factor* and *N-Refinement Factor*, respectively. You can identify the M and N direction by selecting *View* → *Grid* → *Lines and M, N Indices*.

Restriction:

- ◇ The number of refinement must be an integer number.

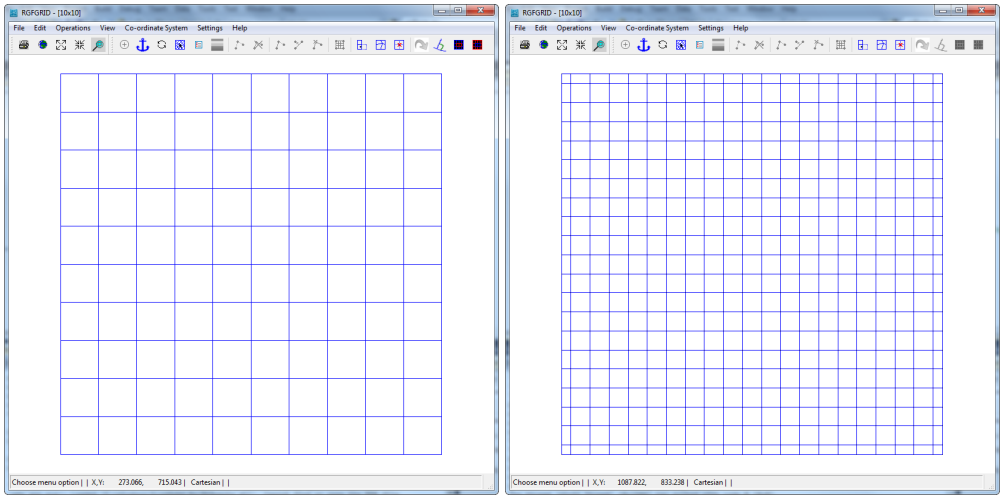
Derefine grid

This option operates on the whole grid and in both directions.

The opposite of *Refine Grid*. One limitation of the refinement procedure is that it can only refine by an integer number. The combination of refine and de-refine allows you to reach a rational number as refinement factor. e.g. You wish a refinement factor of 1.5; first refine by a factor of 3, next de-refine by a factor of 2. Next, go to *Edit* → *Line* → *Smooth* to decrease the jump in grid-sizes.

Casulli

Only applicable for whole irregular squared cell meshes.



(a) *Original square grid ($\Delta x = 100\ m$)* ***(b)*** *After Casulli refinement (most grid cells have $\Delta x = 50\ m$)*

Figure 5.26: *Example of Casulli refinement of an irregular grid*

5.3.11 Orthogonalise grid

This option operates on the whole grid or on a part of the grid. To operate on a part of the grid:

- ◇ For regular grids use *Edit* → *Block* → *Orthogonalise*. The grid will be orthogonalised in accordance with the local grid cell resolution, i.e. the overall shape will be conserved, but individual points may be shifted to get better orthogonality. You can specify parameters that control the orthogonalisation in *Settings* → *Orthogonalisation (regular)*, see Figure 5.44.
- ◇ For irregular grids you have to specify one or more polygons. The orthogonalisation will be performed only for the **selected** polygons. You can specify parameters that control the orthogonalisation in *Settings* → *Orthogonalisation (irregular)*, see Figure 5.45.

5.3.12 Flip edges

Minimalize the number of edges connected to a node. The optimal number of edges to a node is six.

Nodes that are connected to more than, say, six other nodes, are typically enclosed by cells of highly non-uniform shape and wildly varying sizes. This motivates to improve the mesh connectivity by selecting *Operation* → *Flip Edges*.

5.3.13 Grid properties

Specify the desired grid property to be shown, see Figure 5.27.

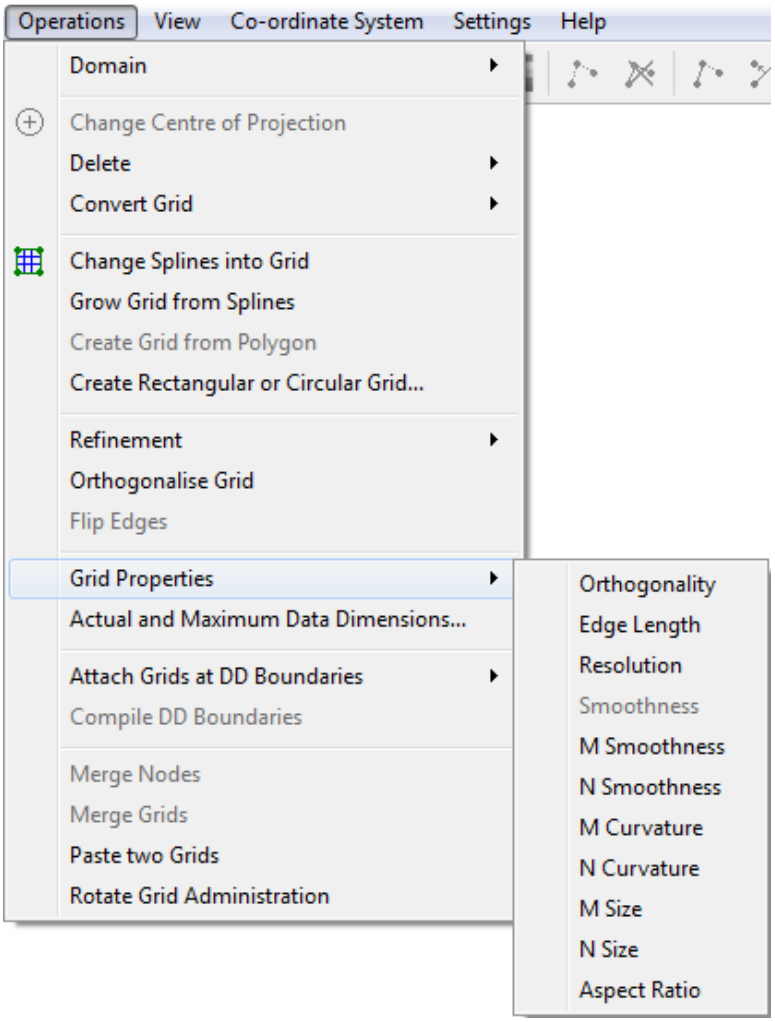


Figure 5.27: Operations → Grid Properties *options*

Orthogonality

Regular grids:
Cell centred cosine value. Keep this value low in the inner model area, e.g. 0.02-0.04. The error in the direction of the pressure gradient in Delft3D-FLOW is proportional to the deviation of the cosine value from zero. Near closed boundaries, larger values can be tolerated than in the inner model area.

Irregular grids:
Cosine value of the angle between an edge and the line between the circumcentres of the enclosing elements of that edge. Keep this value low, e.g. < 0.001.

Edge length

Show the edge length.

Resolution

Square root of grid cell area ([m]).

Smoothness

Irregular grids only. Plotting irregular grid smoothness not yet implemented.

M-Smoothness

Regular grids only. Ratio between adjacent grid cell lengths in M-direction, value ≥ 1. Preferably less than 1.2 in the area of interest.

N-Smoothness

Regular grids only. Ratio between adjacent grid cell lengths in N-direction, value ≥ 1. Preferably less than 1.2 in the area of interest.

M-Curvature

Regular grids only. Reciprocal value of radius of curvature, times 1000 ([1/m]).

N-Curvature

Regular grids only. Reciprocal value of radius of curvature, times 1000 ([1/m]).

M-Size

Regular grids only. Grid cell size in M-direction ([m]).

N-Size

Regular grids only. Grid cell size in N-direction ([m]).

Aspect-Ratio

Regular grids only. Ratio of M-size/N-size, value ≥ 1. Must be in the range [1,2] unless the flow is predominantly along one of the grid lines.

Remark:

- ◇ For a spherical grid the resolution, curvature and grid size are also given in the metric system.

5.3.14 Actual and maximum data dimensions

The actual and maximum dimensions of various data objects are presented in ‘history’, see Figure 5.28

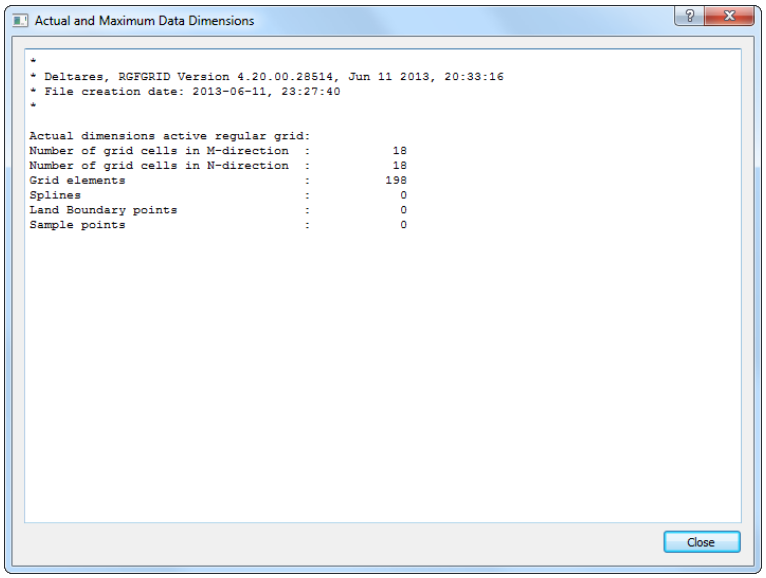


Figure 5.28: Operations *menu,* Actual and Maximum data dimensions

5.3.15 Attach grids at DD boundaries

There is a small difference between attaching regular and irregular grids. For regular grids you are able to move the DD-boundary points of one regular grid to the DD-boundary of the other regular grid, so the boundary is exactly on the same place. After that operation you have to perform the operation *Menu* →*Operations* →*Compile DD Boundaries*. For irregular grids the DD-boundaries should have the same location before the merge operation can be applied by *Operations* →*Attach Grids at DD Boundaries* →*Irregular grids*.

For the menu layout see Figure 5.31.

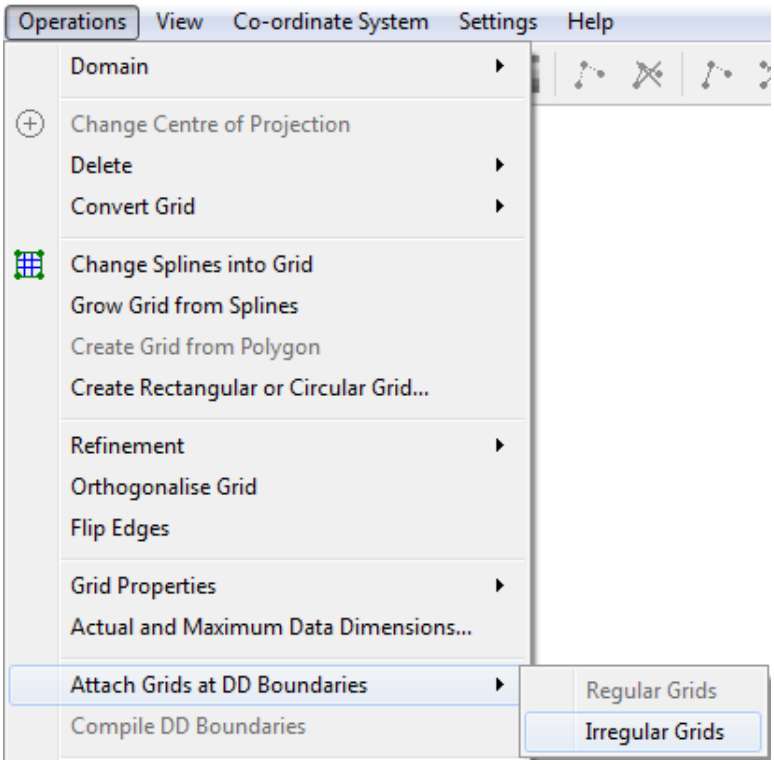
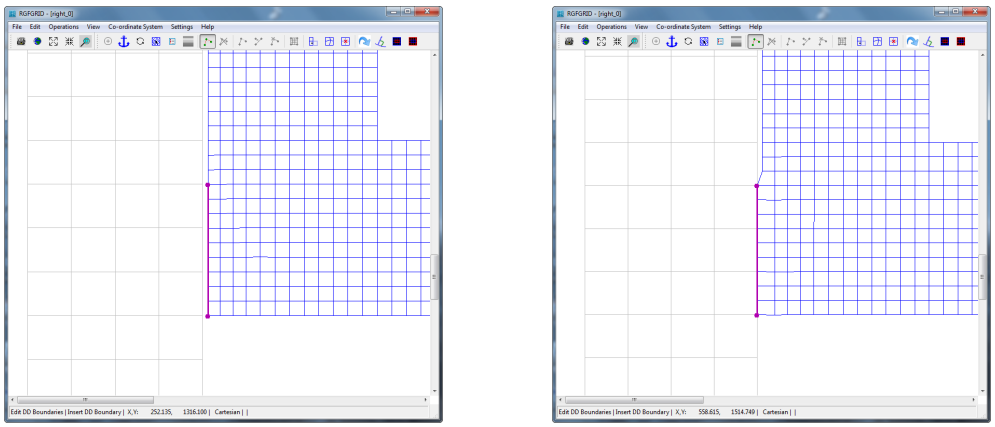


Figure 5.29: Operations →Attach Grids at DD Boundaries

5.3.15.1 Attach regular grids at DD boundaries

This option is only relevant if you want to use the multi-domain option of Delft3D-FLOW. First you have to indicate the domain decomposition boundaries in the mode *Edit → DD Boundaries*.

One of the restrictions of domain decomposition is that the domain boundary between two domains have to coincide, so there is no overlap or gap between the domains on the DD-boundary. This option attach the grid at the DD-boundary to each other, for all DD-boundaries of the current active grid. This is achieved by moving the grid points on the DD-bounday of the active grid, to the corresponding inactive grid, see Figure 5.30.



(a) *Before attach operation* ***(b)*** *After attach operation*

Figure 5.30: Operations → Attach Grids at DD Boundaries→Regular grids

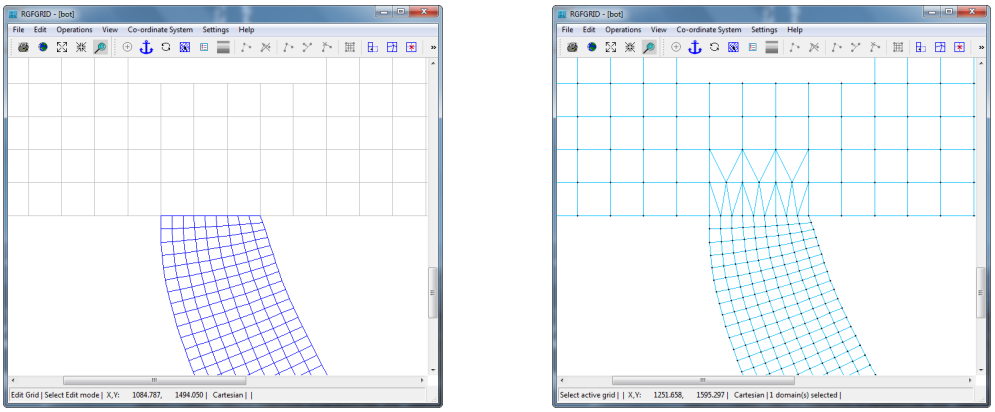
5.3.15.2 Attach irregular grids at DD boundaries

This option is most relevant if you have a multi-domain simulation model suitable for Delft3D-FLOW and you want to use D-Flow FM.

Load the regular grids which need to be merged to one irregular grid at domain decomposition boundaries (ex. Figure 5.31a).

Convert the regular grids to irregular by choosing menu option *Operations* →*Convert Grid* →*Regular to Irregular*.

To perform the merge of the irregular grids choose menu option *Operations* →*Attach Grids at DD Boundaries* →*Irregular grids* (ex. Figure 5.31b).



(a) *Before attach operation* ***(b)*** *After attach operation*

Figure 5.31: Operations → Attach Grids at DD Boundaries

5.3.16 Compile DD boundaries

This option is only relevant if you want to use the multi-domain option of Delft3D-FLOW. First you have to indicate the domain decomposition (DD-)boundaries in the mode *Edit* → *DD Boundaries*.

When you have defined the DD Boundaries, the grids needed in your multi-domain application, are coupled here. Upon clicking *Operations* → *Compile DD Boundaries* a window opens, in which you can select where the DD-boundary will be saved, see Figure 5.32.

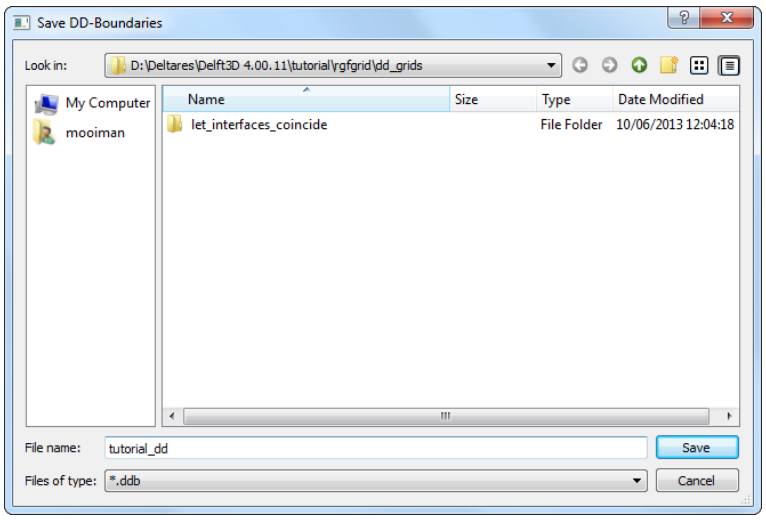


***Figure 5.32: Save DD-Boundaries** window*

The DD administration is written to a file with default mask <∗.ddb>, see Appendix A.9 for its format.

5.3.17 Merge nodes

With this option you are able to merge nodes. First you click the node which you want to merge and than you click the node to merge to. The first clicked node will move to the second node location and than the two nodes will be merged to one node.

5.3.18 Merge grids

To merge several grids you have to indicated which grids need to be merged, to select the grids use the multi selection tool (*Edit* → *Allow Multi Select*). After selecting some grids this option will merge the indicated irregular grids. Nodes from different grids with the same location will be merged.

5.3.19 Paste two grids

The second (inactive) grid is pasted to the active grid. The M, N-orientations of both grids do not have to match. The grid points on the junction line(s) should be relatively close to each other, i.e. less than one quarter of a grid cell apart. On the junction line, the grid points are a weighted average of the active grid and the inactive grid. The weighting factor can be changed in the menu option *Settings* → *General*, see Figre 5.42.

A value of 0.0 will freeze the active grid; the inactive grid will move to the active grid.

5.3.20 Rotate grid administration

The M, N orientation of the grid is rotated over 90 degrees (counter clock-wise). Maybe you need to adjust the grid administration because the grid administration over a DD-boundary should have the same grid orientation.

5.4 View menu

On the *View* menu (see Figure 5.33) options are presented how to display a spherical grid, whether or not to show the boundary and legend, to inspect the grid in 3 dimensions, how to display the objects, and to view grid properties. Display characteristics (for the legend, colours and sizes) may be changed in the *Settings* menu.

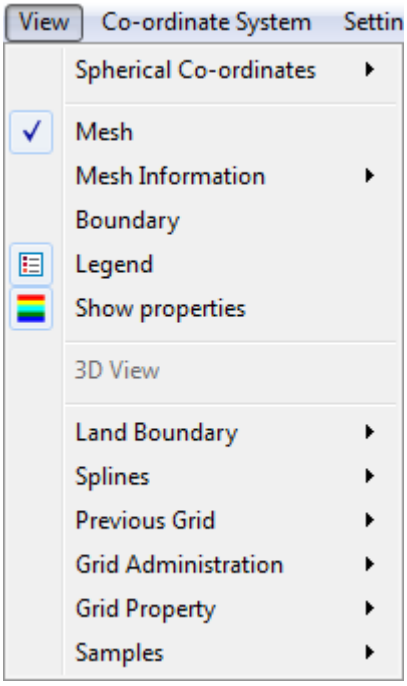


Figure 5.33: *Options on the* View *menu*

5.4.1 Spherical co-ordinates

Default: A spherical grid is shown in stereographic projected co-ordinates.

In the spherical co-ordinate system you can view the objects stereographic projected, see Figure 5.35.

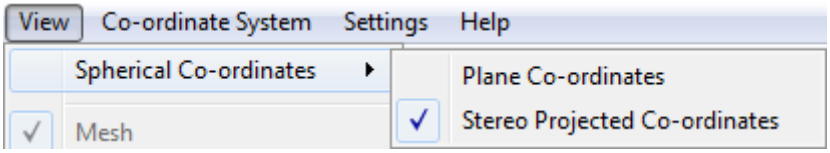


Figure 5.34: *Options on the* View → Spherical Co-ordinates *menu*

Plane co-ordinates

The co-ordinates are displayed just as they are and there is no well known projection used.

Stereographic projected co-ordinates

To display the co-ordinates a stereographic projection is used. The centre point of the stereographic projection is default the centre of the window. When zooming or scrolling the centre point is not recalculated, for recalculation the centre point you have to press menu option *Operations* → *Change Centre of Projection*.

5.4.2 Mesh

Show or hide the meshs

5.4.3 Mesh Information

Upon selecting *View* → *Mesh Information* you show or hide several numebrings of the mesh

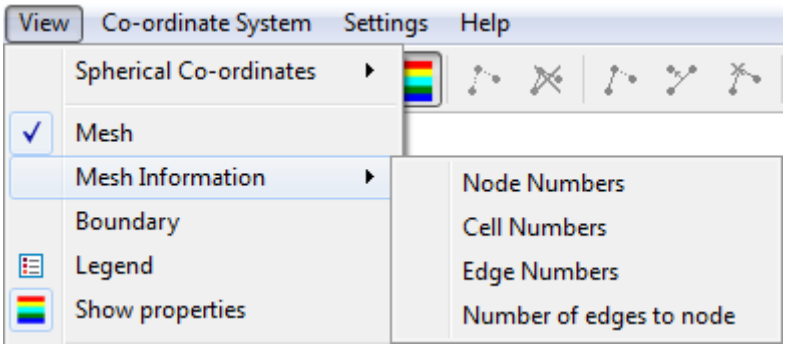


Figure 5.35: *Options on the* View → Mesh Information *menu*

Node numbers

Show or hide the node number.for regular grids it has the format (m, n) and for irregular grids it is a single integer.

Cell numbers

Show or hide the cell number.for regular grids it has the format (m, n) and for irregular grids it is a single integer.

Edge numbers

Show or hide the edge number.

Number of edges to node

Show or hide the number of edges that are connected to a node

5.4.4 Boundary

Default: Hide the boundary.

Show or hide the boundaries, open, domain decomposition as well as computational boundaries

5.4.5 Legend

Default: Show the (colour) legend.

Show or hide the colour legend on the screen.

5.4.6 Show properties

Show or hide the grid properties.

5.4.7 3D View

In the **Delft3D-3DView** window a fully 3-dimensional view of the data is shown.

`c`: Switch rendering mode
`h`: Toggle help
`i`: Inverse depth
`r`: Reste view
`s`: Toggle samples
`x`: Increase depth
`z`: Decrease depth
left mouse: Rotate in xy-plane
`CTRL`+left mouse: move origin xy-plane
middle-mouse: zoom
right mouse: rotate z-axis

5.4.8 Land boundary

Default: Show the land boundary as lines.

The following display options are available for displaying the land boundary:

- ◇ No Land Boundary
- ◇ Lines
- ◇ Filled

5.4.9 Splines

Default: Show the splines as lines with dots.

The following display options are available:

- ◇ No Spline
- ◇ Lines with Dots
- ◇ Lines with Dots and M, N Indices

5.4.10 Grid

Default: Show the grid as lines.

The following options are available for displaying a grid:

- ◇ No Grid
- ◇ Lines
- ◇ Lines and M, N Indices

The latter option is provided so that you can check and control the grid administrative lower left corner, i.e. the location of the (1,1) point. It gives the least confusion if this point is, the more the less, located at the lower left corner of the screen.

5.4.11 Previous grid

Default: Hide the previous grid as lines.

Sometimes, when editing the regular grid, it may be convenient to display the grid both in its present and previous state on the screen at the same time. The usual display options are available:

- ◇ No Previous Grid
- ◇ Lines
- ◇ Lines and M, N Indices

5.4.12 Grid administration

Default: Hide the grid administration.

This option allows you to visualise the grid topology in the 'computational' space (as opposed to the physical space). It helps you decide which grid extensions are allowable so that overlap is avoided. The each domain grid should always have a mono-block structure.

Select the required option:

- ◇ No Grid Administration
- ◇ Lines
- ◇ Lines and M, N Indices

5.4.13 Grid properties

Default: Show the grid property as continuous shading.

Specify how to display the desired grid property:

- ◇ No Grid Property
- ◇ Continuous Shading
- ◇ Coloured Dots
- ◇ Numbers
- ◇ Coloured edges

5.4.14 Samples

Default: Show samples as coloured dots.

Specify how to display the samples:

- ◇ No Samples
- ◇ Coloured Dots
- ◇ Coloured Numbers
- ◇ Mono Coloured Numbers

5.5 Co-ordinate System menu

On the Co-ordinate System menu you can set the desired co-ordinate system (see Figure 5.36) to Cartesian or spherical co-ordinates, see Figure 5.37. Furthermore, you can translate or rotate the objects in a Cartesian co-ordinate system and you can transform Cartesian co-ordinates to spherical co-ordinates and vice versa.

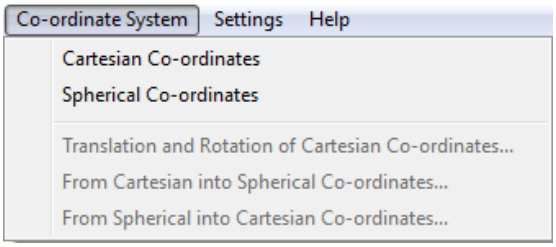
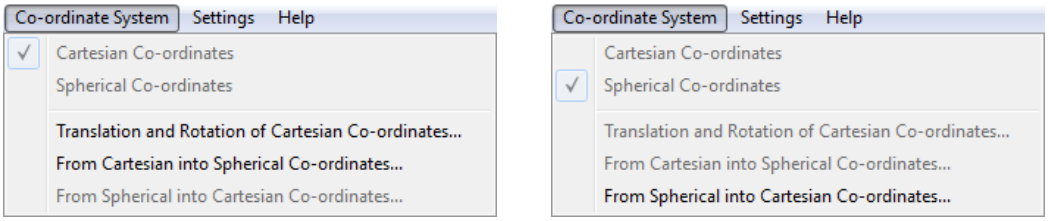


Figure 5.36: *Menu option* Co-ordinate System



(a) *Cartesian co-ordinates selected* ***(b)*** *Spherical co-ordinates selected*

Figure 5.37: *Menu option* Co-ordinate System.

5.5.1 Cartesian co-ordinates

In this case the co-ordinates are easting and northing in metres.

5.5.2 Spherical co-ordinates

In this case the co-ordinates are longitude and latitude, in decimal degrees.

5.5.3 Translation and rotation of Cartesian co-ordinates

This option may be applied if you are changing to a new Cartesian co-ordinate system which has a different position of the origin or another orientation. The parameters involved are, see Figure 5.38:

- ◇ Offset X direction [m] default 0.0
- ◇ Offset Y direction [m] default 0.0
- ◇ Rotation left [degrees] default 0.0
- ◇ X Scale factor default 1.0
- ◇ Y Scale factor default 1.0

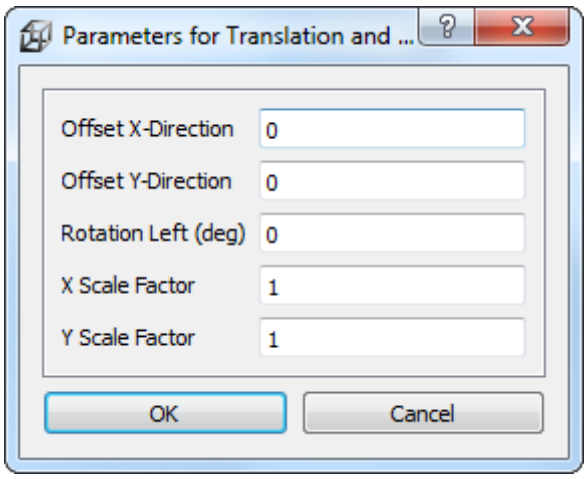


Figure 5.38: Parameters for translation and rotation *form for transformation to Cartesian co-ordinates*

Remark:

- ◇ A translation and rotation operates only on samples, polygon the active grid.

5.5.4 From Cartesian into spherical co-ordinates

A form will appear with the parameters for the co-ordinate conversion, see Figure 5.39. The first parameter indicates the current co-ordinate system:

1 UTM,
2 Amersfoort / RD new (EPSG:28992) or
3 RD (Parijs) (EPSG:2489).

Putting the cursor on top of this field will show the help text. If the current system is UTM you

have to specify the zone number

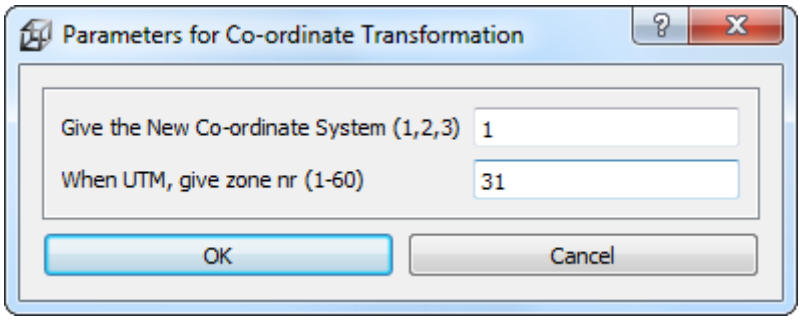


Figure 5.39: Parameters for Co-ordinate transformation *form for transformation to spherical co-ordinates*

5.5.5 From spherical into Cartesion co-ordinates

Figure 5.40 shows the parameters involved for the conversion from spherical to Cartesian co-ordinates.

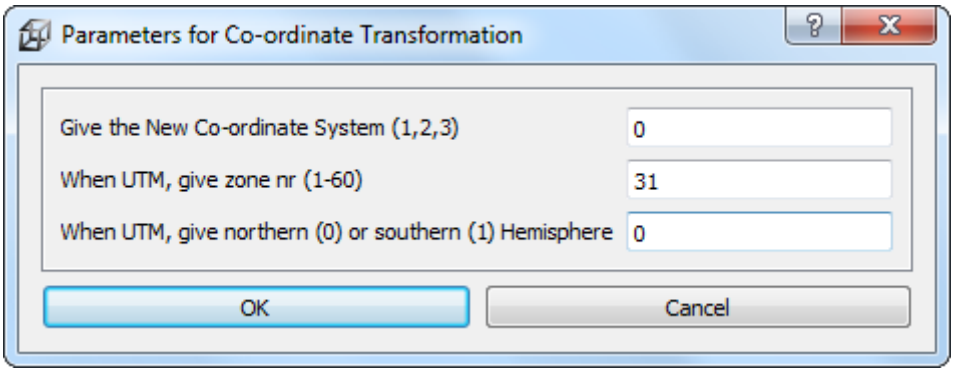


Figure 5.40: Parameters for Co-ordinate transformation *form for transformation to Cartesian co-ordinates*

5.6 Settings menu

The following options can be accessed through the *Settings* menu, see Figure 5.41

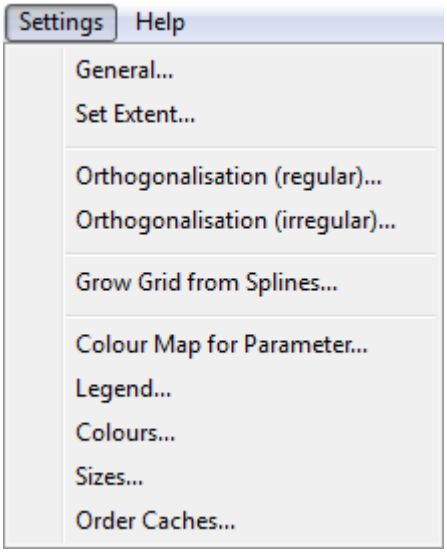


Figure 5.41: *Options on* Settings *menu*

5.6.1 General

The following parameters influence the behaviour of the operations above. They are set via the following parameter list, see Figure 5.42

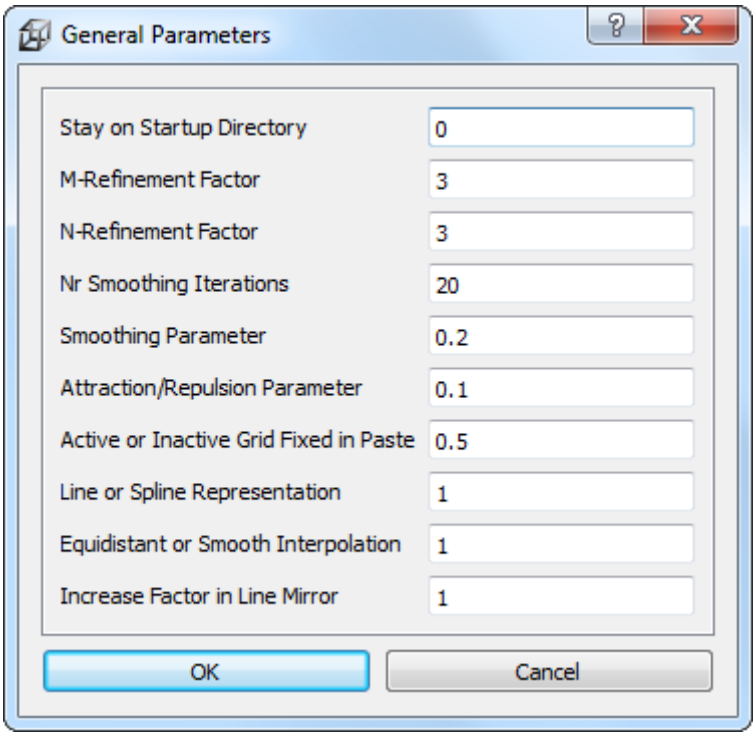


Figure 5.42: *Options on* ***Settings*** *window*

- ◇ Stay on Startup Directory default: 0 (Off)
  When navigating through the directories in the file menu, you can specify whether to keep the latest visited directory (0), or always go back to the start-up directory (1).
- ◇ M-refinement factor default: 3
  A value of 2 gives twice as many grid cells in m-direction. An odd value assure that the

cell centre and the mid of a cell edge are available in the coarse as well as in the refined grid.

- ◇ N-refinement factor default: 3
  A value of 2 gives twice as many grid cells in n-direction. An odd value assure that the cell centre and the mid of a cell edge are available in the coarse as well as in the refined grid.
- ◇ Nr Smoothing Iterations default: 20
  The smoothing in edit mode is controlled by this parameter.
- ◇ Smoothing Parameter default: 0.2
  The smoothing in edit mode is also controlled by this parameter. A value of 0.0 results in no smoothing, a value of 1.0 in maximum smoothing.
- ◇ Attraction/Repulsion Parameter default: 0.1
  Attraction/repulsion in edit mode is controlled by this parameter. The value is the fractional change in size of the first grid cell adjacent to the indicated line. Increase this value for more attraction or repulsion.
- ◇ Active or Inactive Grid Fixed in Paste default: 0.5
  When pasting an active grid to an inactive grid, the grid points on the grid junction line are a weighted average between both grids. If you want to keep these points in the position of the active grid, set this parameter to 0.0. To keep the position of the inactive grid, choose 1.0; a value in between averages.
- ◇ Line or Spline Representation (0 or 1) default: 1 (Spline)
  Splines, or grid boundaries in the orthogonalisation process, can also be represented as straight lines if this parameter is put to a zero value.
- ◇ Equidistant or Smooth Interpolation (0 or 1) default: 1 (Smooth)
  When interpolating the splines into a grid, equidistant interpolation can be specified using a value of 0.
- ◇ Increase Factor in Line Mirror (1.0 - 10.0) default: 1.0
  When adding grid cells using the *Edit* → *Line* → *Mirror* option, this parameter defines the size of the new grid cells.

5.6.2 Set extent

Set the horizontal extent of the canvas

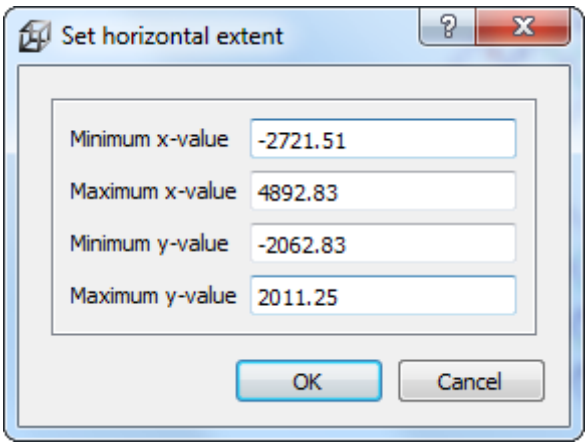


***Figure 5.43: Set horizontal extent** window*

- ◇ Minimum visible x-value
- ◇ Maximum visible x-value

- ◇ Minimum visible y-value
- ◇ Maximum visible y-value

The used extent is dependent on the current window size, keeping the x/y ratio to 1.

5.6.3 Orthogonalisation regular

With the **Orthogonalisation Parameters** form, see Figure 5.44, the orthogonalisation process can be controlled:

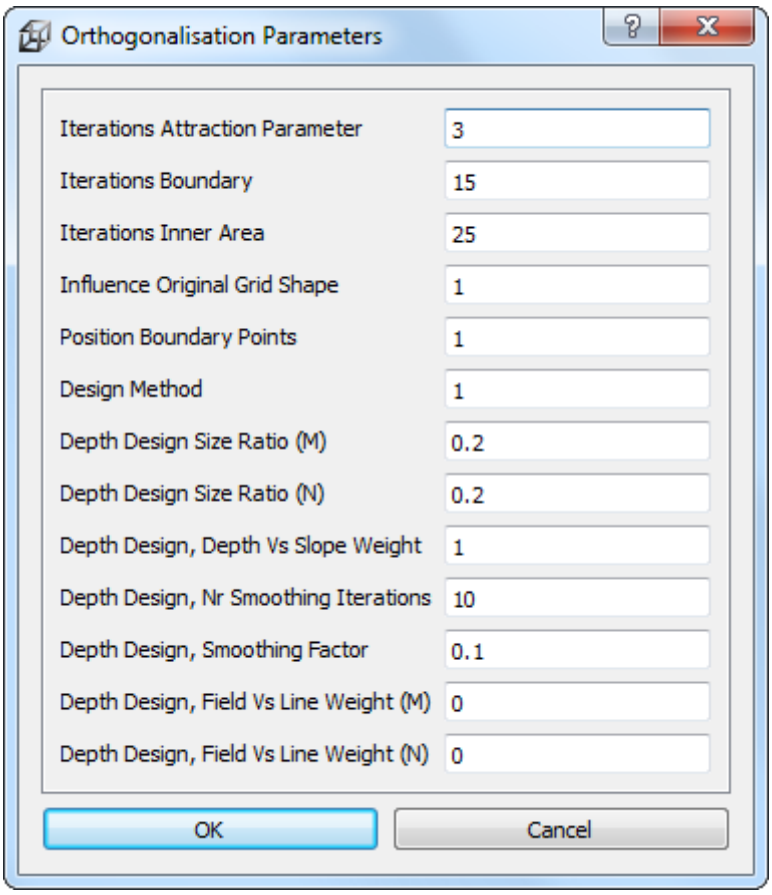


Figure 5.44: *Options on* ***Orthogonalisation Parameters*** *window*

- ◇ Iterations Attraction Parameter default: 3
  The shape of the resulting grid is based on the so-called attraction parameter, i.e. the local aspect ratio of the original grid. One complete orthogonalisation cycle consists of three loops. The outer loop is the attraction parameter loop, in which this parameter field is established. Only few of these loops are usually performed. Next, several boundary loops are performed, in each of which the inner area is solved several times. Increasing the number of attraction parameter iterations improves orthogonality, but it increases deviation from the originally designed shape.
- ◇ Iterations Boundary default: 15
  In one boundary loop, all boundary points are updated once, and all inner area points are updated as many times as specified by the next parameter. We advise values in the following range: [5 - 20].
- ◇ Iterations Inner Area default: 25

The number of inner area iterations in the orthogonalisation is advised in the range: [10 - 50].

◇ Influence Original Grid Shape default: 1.0
This parameter specifies the influence of the specified grid shape in the inner area during the orthogonalisation procedure. The grid shape in the inner area can be specified in three ways, see 'Design Method' below. With a value of 1 the specified shape is maintained as closely as possible. With a value of 0, the shape mostly depends on the shape of the boundaries and the internal corner points. Any value between 0 and 1 can be chosen.

◇ Position Boundary Points default: 1.0
This parameter specifies the freedom of movement of boundary points. These points move along splines spanned by the outer points of the grid. A value of 1 gives full freedom of movement, whereas a value of 0 keeps boundary points completely fixed. Any value between 0 and 1 may be chosen.

◇ Design Method, 1, 2, or 3 default: 1
This parameter specifies in what way the attraction parameter (local aspect ratio) field is created. The three methods are:

1 This method is based upon the aspect ratios of the original design grid (default).
2 This method uses a polygon, the polygon can be applied to control grid spacing.
3 With this method the grid resolution is controlled based upon features in a bathymetry that can be opened by means of the samples. This method is therefore called 'Depth design'.

In these methods, the attraction parameter field is based upon grid spacing functions both in the M- and N-Direction. Their local ratio forms the desired attraction parameter field. Both the M- and N-grid spacing functions can be controlled by a number of parameters, that are explained below. The same parameters also apply to method 2, that can be seen as a special case of method 3, in which the bathymetry is specified by specifying a constant 'depth' inside the polygon, different from the also constant 'depth' outside the polygon.

◇ Depth Design Size Ratio (M) default: 0.2
Both for the M- and the N-direction, the size ratio between the smallest and largest grid size in that direction can be specified. If a value of 1 is specified, a uniform distribution results. Choosing a small value will result in large grid size variation. If a value of 0 is specified, this is seen as a special case and the original grid shape is applied as the desired grid spacing function.

◇ Depth Design Size Ratio (N) default: 0.2
See *Depth Design Size Ratio (M)* above.

◇ Depth Design, Depth vs Slope Weight default: 1
Both the depth and the slope can be applied as grid spacing control functions. To obtain a high resolution in deep areas only, specify a *Small/large Size Ratio* below 1, e.g. 0.1, and specify a *Depth vs. Slope Weight* parameter of 1. To obtain small cells at steep slopes only, specify a value of 0. Any value in between 0 and 1 can be applied. In the future, the slope variation is foreseen as a controlling parameter as well.

◇ Depth Design, Number of Smoothing Iterations default: 10
To obtain smoother transitions between sloping and non-sloping areas, the grid spacing functions can be smoothed. The smoothing parameters apply both to the (M) and (N) direction. Also, the smoothing may be applied to 'spread' the grid spacing information towards grid cells that initially may lie outside the area that needs to get a high resolution. In each attraction parameter iteration, see above, the grid point spacing function is evaluated, and applied in the following orthogonalisation loop, which results in shifting the grid

points to their final position. Once getting closer to their final position, the smoothing may be decreased, so that the bathymetry features become more apparent in the grid. This process may be automated in future.

◇ Depth Design, Smoothing Factor default: 0.1
Smoothing weight of point itself to neighbours.

◇ Depth Design, Field vs. Line Weight (M) default: 0
The *Small/Large Size Ratio* parameter can either be specified to apply to the whole grid, or to every grid line. i.e. should the specified *Size Ratio* in the given direction be applied to the whole grid or to every grid line? If a value of 1 is chosen, this ratio will only occur at the maximum value of entire spacing function. If a value of 0 is chosen, this size ration will occur at every grid line.

◇ Depth Design, Field vs. Line Weight (N) default: 0
See *Depth Design, Field vs. Line Weight (N)* above.

5.6.4 Orthogonalisation irregular

With the **Orthogonalisation Parameters (irregular)** form, see Figure 5.45, the orthogonalisation process of irregular grids can be controlled:

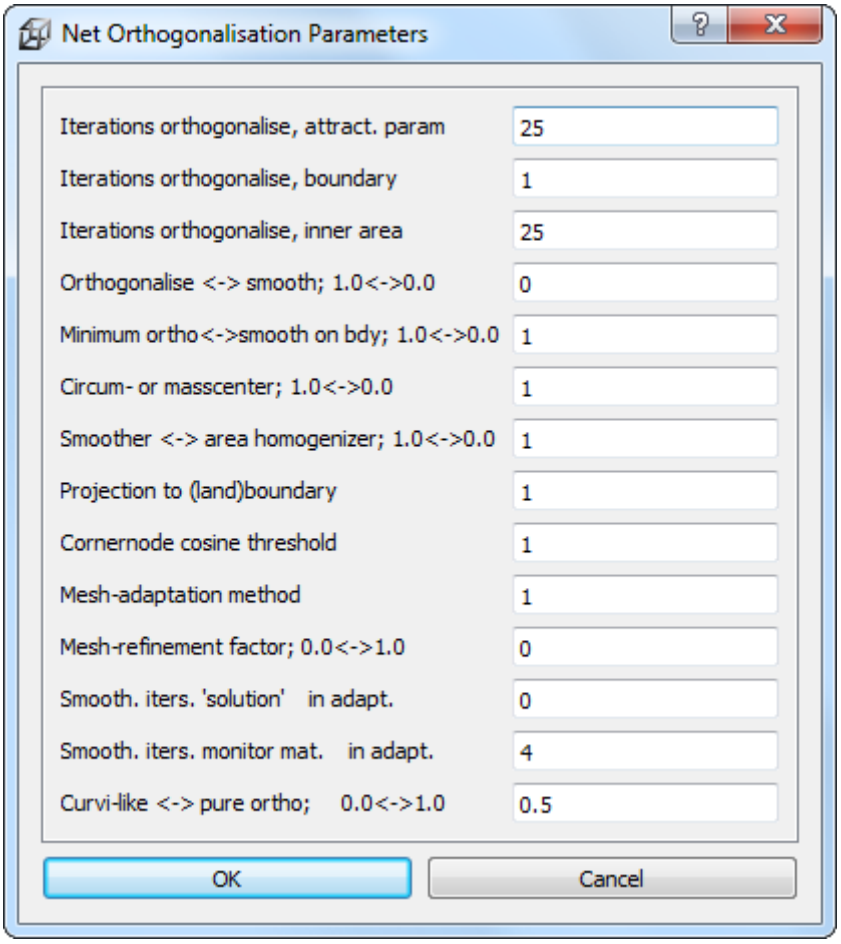


Figure 5.45: *Options on* ***Orthogonalisation Parameters (irregular)*** *window*

◇ Iterations orthogonalise, attraction parameters: Default: 25
The number of iterations in which the attraction parameters are computed for the grid. The attraction parameter is the fractional change in size of the first grid cell adjacent to the

indicated line.

◇ Iterations orthogonalise, boundary: Default: 1
This parameter can be used to prescribe the number of iterations in which the grid is moved along the boundaries to improve the orthogonality of the grid.

◇ Iterations orthogonalise, inner area: Default: 25
This parameter can be used to prescribe the number of iterations in which the grid is moved along in the interior of the domain to improve the orthogonality of the grid. The total number of iterations is the product of the three iteration values (attract. param, boundary and inner area).

◇ Orthogonalise $\leftrightarrow$ smooth; 1.0 $\leftrightarrow$ 0.0 Default: 0
The balance between mesh-smoothing (0.0) and mesh-orthogonalization (1.0). One has to keep in mind that mesh smoothing (ortho. param. $\rightarrow$ 0) will compromise mesh orthogonality. Sole orthogonalization (ortho. parameter=1) on the other hand, can cause highly distorted, non-smooth meshes, especially for meshes consisting of quadrilaterals. It is advised to choose a low orthogonalization parameter and repeat the process while gradually increasing the orthogonalization parameter at every repetition.

stage	1	2	3	4	5	6
ortho. parameter	0.5	0.8	0.9	0.99	0.999	0.9999

Table 5.1: *Multi-stage orthogonalization strategy*

◇ Minimum ortho $\leftrightarrow$ smooth on boundary; 1.0 $\leftrightarrow$ 0.0 Default: 1
This parameter can be used to prescribe the number of iterations in which the grid is moved along in the interior of the domain to improve the orthogonality of the grid. The total number of iterations is the product of the three iteration values (attract. param, boundary and inner area).

◇ Orthogonalise $\leftrightarrow$ smooth; 1.0 $\leftrightarrow$ 0.0 Default: 1
Balance between orthogonalisation and Laplacian smoothing. The orthogonality is defined by the angle between the line connecting water level points and the line connecting two grid cell corners. The smoothness is defined as the ratio between the areas of two adjacent grid cells.

◇ Circum- or masscentre; 1.0 $\leftrightarrow$ 0.0: Default: 1
Define whether the orthogonality is measured based on the circumcenter of the triangle or on the mass center of a triangle.

◇ Smoother $\leftrightarrow$ area homogenizer; 1.0 $\leftrightarrow$ 0.0: Default: 1

◇ Projection to (land)boundary: Default: 1
0 means no projection of the grid to the landboundary while orthogonalising the grid. 1 means projection of the grid boundaries to the original grid boundaries as before orthogonalising the grid. 2 means projection of the grid boundaries to the nearest landboundary. 3 means projection of the grid boundaries as well as parts of interior part of the grid to the nearest landboundaries.

◇ Corner node cosine threshold: Default: 1
Determines whether a node is a corner on the basis of the cosine of the boundary edge angle. If a node is a corner, then the node is not moved during orthogonalisation.

◇ Mesh adaption method: Default: 1
Selection of a mesh-adaptation method. 0 means a Winslow type monitor function, 1 an arc-length monitor function and 2 a harmonic map monitor function. See Huang (2001, sect 3.3).

◇ Mesh refinement factor; 0.0 $\leftrightarrow$ 1.0: Default: 0

Concentation of the mesh in a refined region (parameter in the mesh-adaptation method). See Huang (2001, sect 3.3).

- ◇ Smooth. iters. 'solution' in adapt.: Default: 0
  Number of smoothing iterations of solution u in mesh-adaptation method. See Huang (2001, sect 3.3).
- ◇ Smooth. iters. 'monitor mat.: Default: 4
  Number of smoothing iterations of monitor matrix G in mesh-adaptation method. See Huang (2001, sect 3.3).
- ◇ Curvi-like ↔ pure ortho; 0.0 ↔ 0.5: Default: 0
  Chooses between pure orthogonalisation versus curvi-grid-like orthogonalisation in quads.

5.6.5 Grow grid from spline

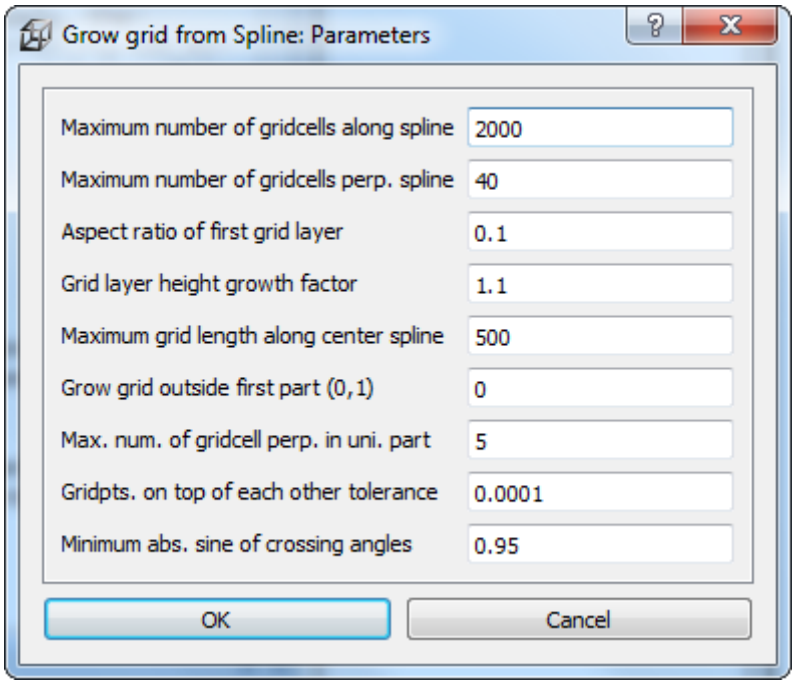


Figure 5.46: *Options on* ***Grow Grid from Spline: Parameters*** *window*

- ◇ Maximum number of grid cells along spline: Default: 2000
  Upper bound of the number of grid cells along the centre spline. The actual number of grid cells is determined by the cell and spline lengths, but will not exceed this number. In practice one has to set this number sufficiently large.
- ◇ Maximum number of grid cells perp. spline: Default: 40
  Upper bound of the number of grid layers that will be grown from the centre spline. The actual number of grid layer is determined by the grid height (specified by the splines) the height of the first grid layer and the grid layer growth factor, see below, but it will not exceed this number. In practice one has to set this number sufficiently large.
- ◇ Aspect ratio of first grid layer: Default: 0.1
  The ratio of the grid cell height and length at the centre spline. If a centre spline is provided solely, the aspect ratio of the grid on either side of the centre spline is determined by this variable.
- ◇ Grid layer height growth factor: Default: 1.1
  The fractional increase of grid layer heights in the exponentially growing part of the grid.
- ◇ Maximum grid length along centre spline: Default: 500

The maximum grid cell length. Note that the length decreases where the spline curvature increases

- ◇ Grow grid outside first part (0,1): Default: 0
  create the exponentially growing grid supplementary to the uniform part (1) or not (0). This parameter has no effect if no uniform part is present, i.e. no bounding splines are provided. In that case the exponentially growing grid is the sole grid created.
- ◇ Maximum number of grid cell perpendicular to the center spline in the uniform part: Default: 5
  The number of grid cells will not exceed this number. If necessary, the cells will be enlarged, and the aspect ratio is disregarded.
- ◇ Gridpoints on top of each other tolerance: Default: 0.0001
  A tolerance on merged grid lines; for expert users only.
- ◇ Minimum absolute sine of crossing angles: Default: 0.95
  Minimum value of $|\sin\alpha|$ where α is the angle between the edge and the line connecting the circumcentres of the adjacent cells to that edge.

5.6.6 Change colour map

When clicking on the *Settings* → *Change Colour Map* menu, a form opens in which you can select the relation between a parameter (i.e. Depth) and the loaded colour maps; see Figure 5.47

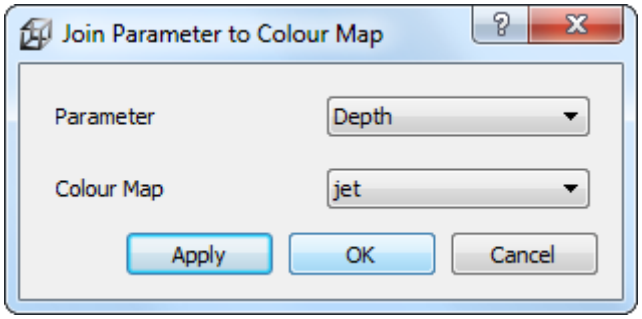


Figure 5.47: *Options on* ***Colour Map for Parameter*** *window*

5.6.7 Legend

When clicking on the *Settings* → *Legend* menu, a form opens in which you can define how the iso-colour figures should be displayed; see Figure 5.48

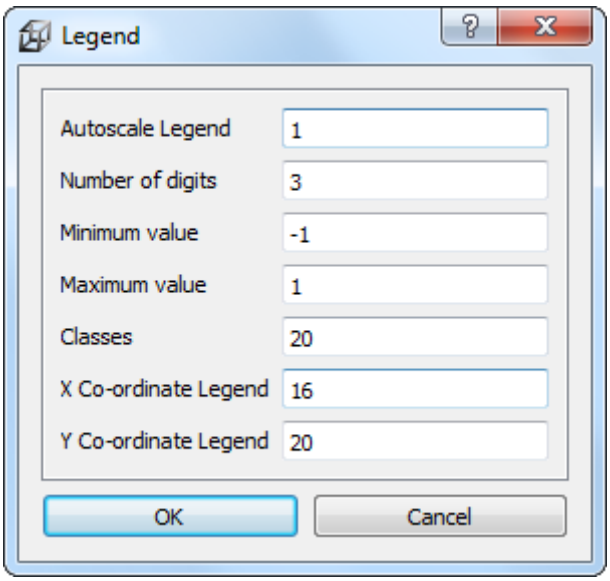


Figure 5.48: *Options on* Settings → Legend *menu*

- ◇ *Autoscale Legend* default: On
  Specify whether the program should determine the iso-colour values automatically, or to do it yourself. If you leave it to the program, it will determine the minimum and maximum depth value within the viewing area and display the number of iso-colours specified above. Zooming in will always result in display of the same number of iso-colours. If you want to specify the iso-colour values yourself, you have to specify one of the three parameters below. When zooming in, the iso-colour values will remain fixed.
- ◇ *Minimum Value* default: Off
  Specifying this value turns autoscale off.
- ◇ *Maximum Value* default: Off
  Specifying this value turns autoscale off.
- ◇ *Classes* default: 20
  The number of classes can be specified
- ◇ *X Co-ordinate Legend* default: 16
  x Co-ordinate of lower left corner of legend in pixels
- ◇ *Y Co-ordinate Legend* default: 20
  y Co-ordinate of lower left corner of legend in pixels

5.6.8 Colours

When clicking on the *Settings* → *Colours* menu, a form opens in which you can define the colours for background, land boundary, polygons, etc.; see Figure 5.49

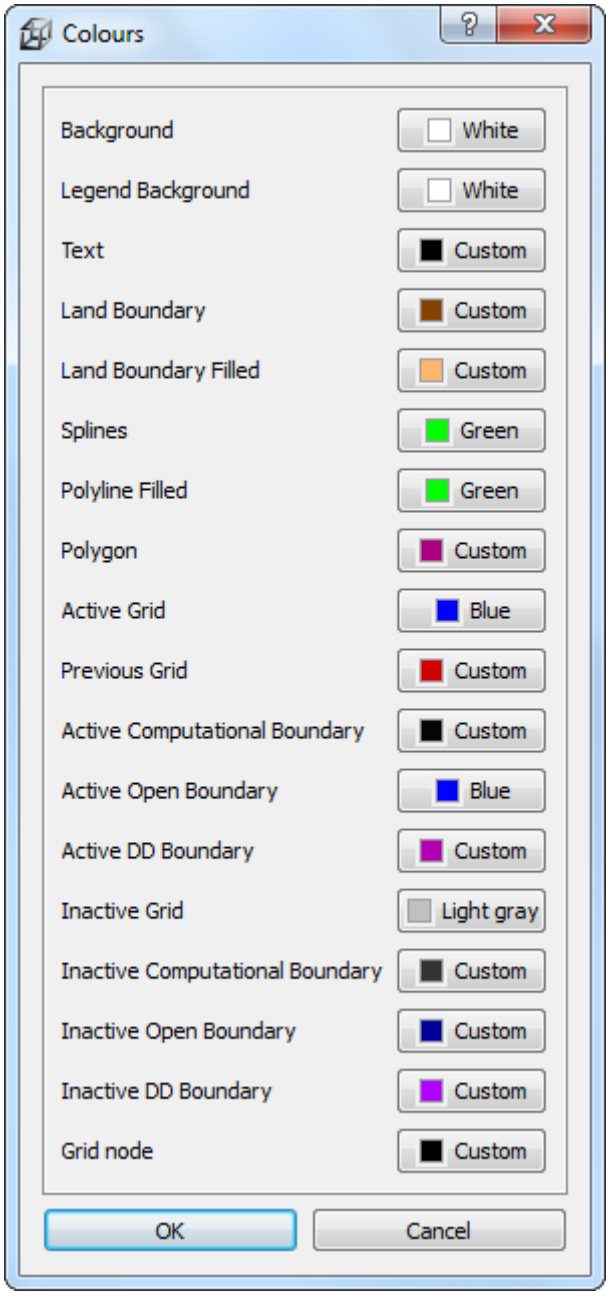


Figure 5.49: *Options on* Settings → Colours *menu*

5.6.9 Sizes

When clicking on the *Settings* → *Sizes* menu, a form opens in which you can define the linewidth and dotsize in pixels. See Figure 5.50

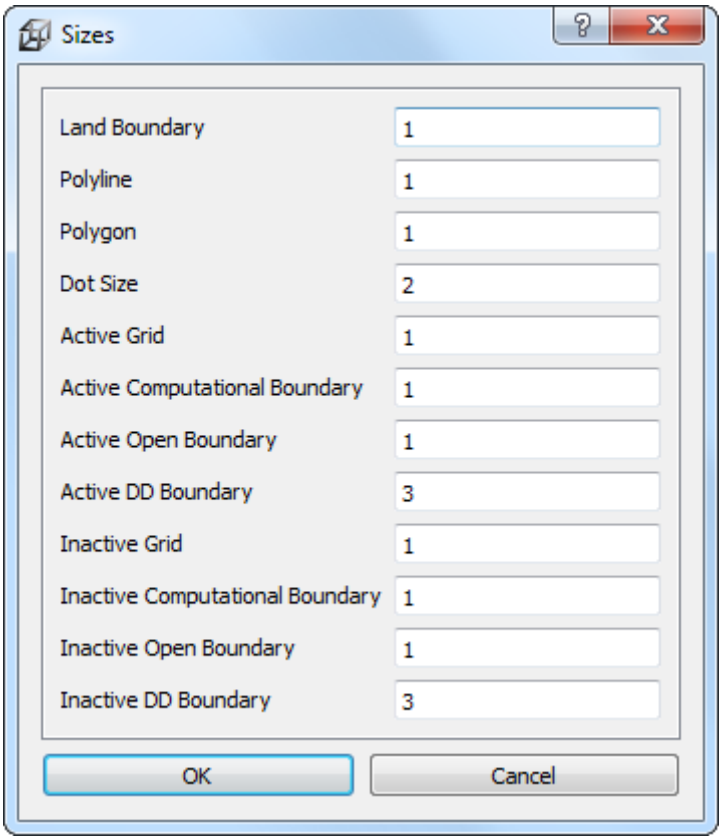


Figure 5.50: *Options on* Settings → Sizes *menu*

5.6.10 Order caches

The parameters set in the **Order caches** window, see Figure 5.51, influence the drawing order of the several items. The drawing order of the caches is: $5, 4, 3, 2, 1, 0$. Cache 5 is drawn first and cache 0 is drawn last. So the items which will drawn in cache 0 are drawn on top. If there is no need to draw a cache it will not be done, this improves the drawing performance by avoiding unnecessary drawings. Therefore, if an item is changed in cache 3 only caches $3, 2, 1$ and 0 are drawn.

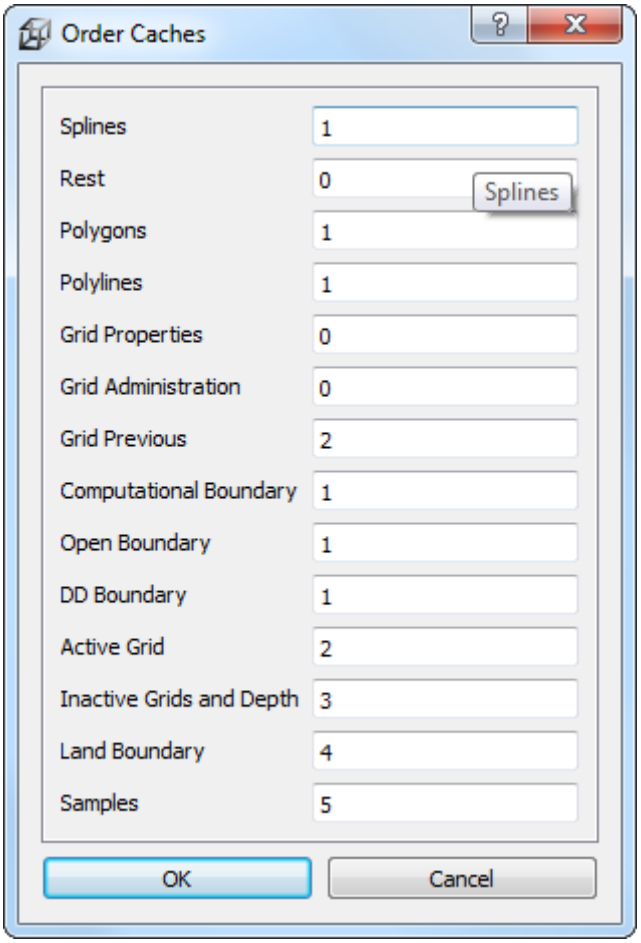


Figure 5.51: *Options on* ***Order Caches*** *window*

- ◇ Splines default: 0
- ◇ Rest default: 0
- ◇ Polygons default: 0
- ◇ Grid Properties default: 0
- ◇ Grid Administration default: 0
- ◇ Grid Previous default: 2
- ◇ Computational Boundary default: 1
- ◇ Open Boundary default: 1
- ◇ DD Boundary default: 1
- ◇ Active Grid default: 2
- ◇ Inactive Grids and Depth default: 3
- ◇ Land Boundary default: 4
- ◇ Samples default: 5

5.7 Help menu

On the *Help* menu, you may choose to read the user manual or the version number of RGFGRID ; see Figure 5.52

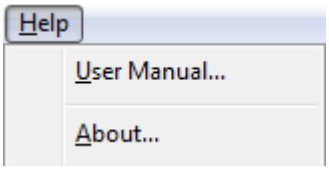


Figure 5.52: *Options on* Help *menu*

5.7.1 User manual

When clicking on the *Help* → *User Manual* the user manual of RGFGRID will be displayed (file <RGFGRID _User_manual.pdf>).

5.7.2 About

When clicking on the *Help* → *About* a window will display the current version number of RGFGRID.

6 Tutorial

6.1 Harbour

Start RGFGRID as explained in Section 3.3 with the current directory set to <$D3D_HOME/tutorial/rgfgrid/harbour>.

This tutorial uses the land boundary and spline files which are available in that directory.

6.1.1 Co-ordinate system

Before opening a land boundary and generating a grid, you have to indicate in which co-ordinate system you are going to work. This example will use Cartesian co-ordinates.

◇ On the *Co-ordinate System* menu, click *Cartesian Co-ordinates*.

6.1.2 Open a land boundary

Open a file which defines the land boundary.

◇ On the *File* menu, point to *Attribute Files* and click *Open Land Boundary*.

If you have not started RGFGRID with the current directory being
<$D3D_HOME/tutorial/rgfgrid/harbour>
then browse to this directory.

◇ Highlight and *Open* file <harbour.ldb>.

After opening the boundary outline as shown in Figure 6.1 is now visible on your screen.

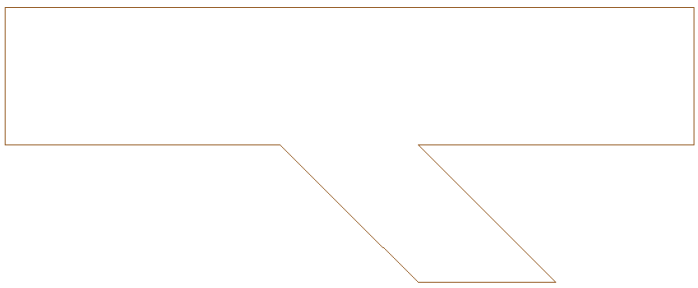


Figure 6.1: *Land boundary outline of <harbour.ldb>*

6.1.3 Zoom in and out

To zoom in or out several facilities are available:

- ◇ click  to zoom in or  to zoom out
- ◇ press the + or - key while keeping the `Ctrl`-key pressed.
- ◇ use the mouse scroll wheel.

To zoom in on a specific area:

- ◇ use  and drag a box around the area.

To zoom out to the full extent:

- ◇ click  to zoom to full extent

To pane the grid: keep the `Ctrl`-key pressed and move around with the cursor.

6.1.4 Define splines

Open a file with definition of splines.

- ◇ On the *File* menu, point to *Import* and click *Splines*.
- ◇ Open spline file <harbour.spl>.

After opening the file with spline definitions, your screen looks like Figure 6.2.

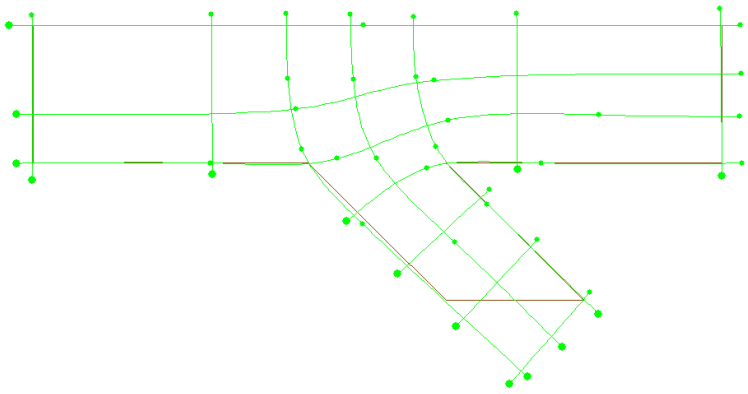


Figure 6.2: *Display of splines and land boundary in the 'harbour' tutorial*

6.1.5 Generate grid from splines

Generating a grid after opening splines.

- ◇ On the *Operations* menu, click *Change Splines into Grid*, or click  on the toolbar.

From the splines a regular grid is generated, see Figure 6.3.

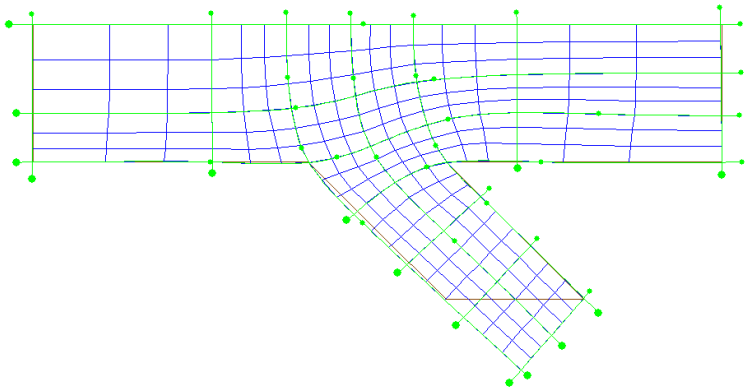


Figure 6.3: *Spline grid changed into result grid with a refinement of 3*

The refinement factors can be specified in *Settings* → *General*. The default is 3 in both directions.

Do not show the spline anymore

◇ On the *View* menu, point to *Splines*, and click *No Splines*, see Figure 6.4.

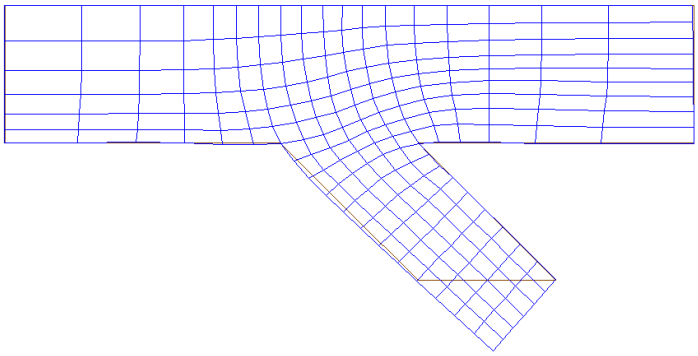


Figure 6.4: *Splines not displayed anymore*

6.1.6 Refine grid

Refine a grid.

◇ On the *Settings* menu, click *General*.
◇ Specify M and N refinement factors of “3” by “3”.
◇ On the *Operations* menu, click *Refine Grid*.

The result of the refinement should look like in Figure 6.5.

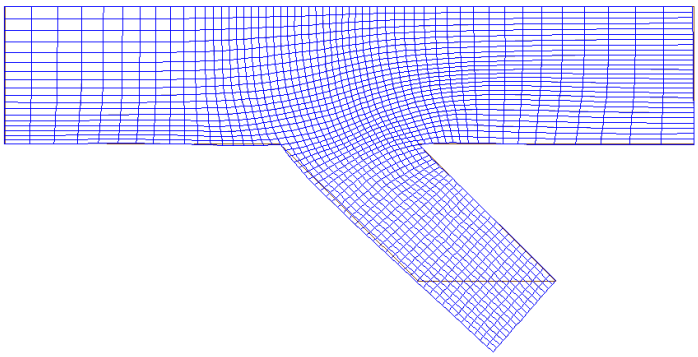


Figure 6.5: *Grid after another refinement of 3 by 3*

6.1.7 Fit grid boundary to land boundary

The grid boundaries can be fitted to the land boundary.

- ◇ On the *Edit* menu, point to *Line* and click *Line to Land Boundary*.
- ◇ You are now in “line to land boundary”.
- ◇ Click the end points of the grid line segment that you want to attach to the land boundary. The concerning segment of the grid line will be highlighted. To expand the area of influence of the attachment transformation click one point on the grid side of the indicated line and perform the transformation, see Figure 6.6. When no influence area is indicated just the grid boundary segment is shifted to the land boundary.

For instance a point halfway in between fixed boundaries.

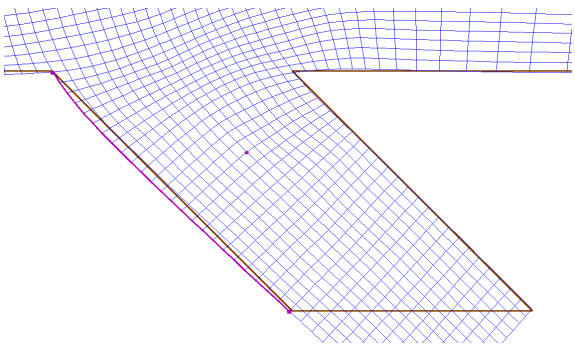


Figure 6.6: *Indicating outer grid line and influence area to be moved to land boundary*

- ◇ Click right to execute the *Line to Land Boundary* action, the result will look like Figure 6.7.

Remark:

- ◇ The previous grid can be shown using one of the options in *View* → *Previous Grid*.

This steps can be repeated until all necessary grid boundaries are fitted to the land boundary.

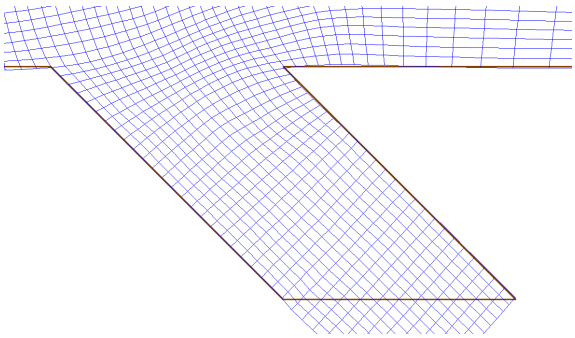


Figure 6.7: *Grid after Line to Land Boundary action*

6.1.8 Check grid orthogonality

To inspect the quality of the grid, for instance the orthogonality:

◇ On the *Operations* menu, point to *Grid Properties*, and click *Orthogonality*.

This will show a plot (see Figure 6.8) of the cosine values of grid corners. The cosine values should be close to zero. The error in the computed direction of the pressure term in Delft3D-FLOW is proportional to these values. In offshore areas the orthogonality should be less than 0.02. Near closed boundaries, higher values are sometimes acceptable.

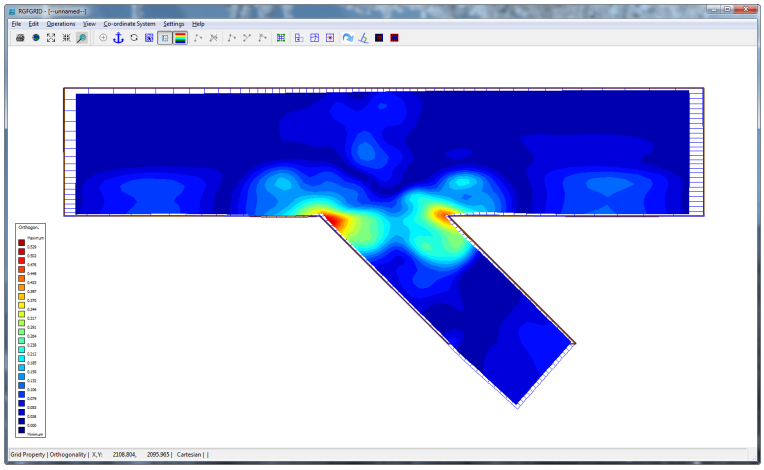


Figure 6.8: *Grid properties; orthogonality*

6.1.9 Orthogonalise grid

Now we will improve the orthogonality:

◇ On the *Operations* menu, click *Orthogonalise Grid* to improve the grid orthogonality.

Remarks:

◇ Default the legend uses auto-scaling. With respect to Figure 6.8, the colours used in Figure 6.9 are the same but the range of cosine values is different.

◇ To set the auto-scaling off, click on *Settings* → *Legend*, and change the appropriate parameter to 0.

To hide the grid properties.

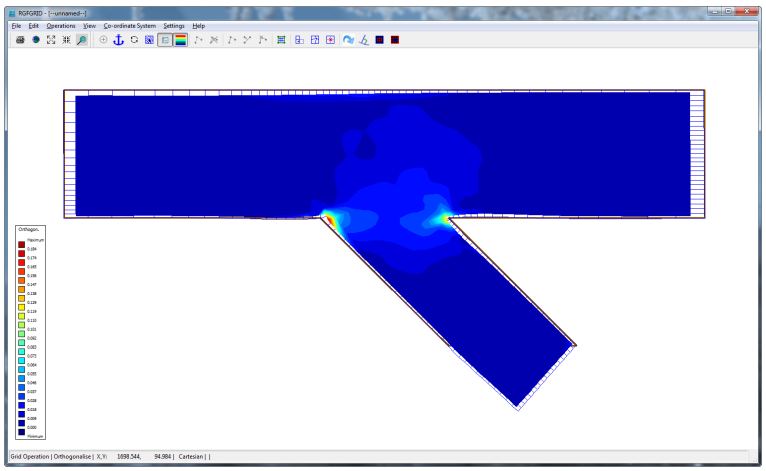


Figure 6.9: *Grid properties; orthogonality. After 1 orthogonalisation action*

◇ Select the *View* menu, point to *Grid Property* and click *No Grid Property*.

In practice, making the grid orthogonal, you will work in blocks rather than on the whole grid. To make the grid orthogonal locally:

◇ Select on the *Edit* menu, point to *Block* and click *Orthogonalise.*
◇ Click 2, 3, or 4 opposite block corners, see Figure 6.10.
◇ Click right to activate the orthogonalisation process.

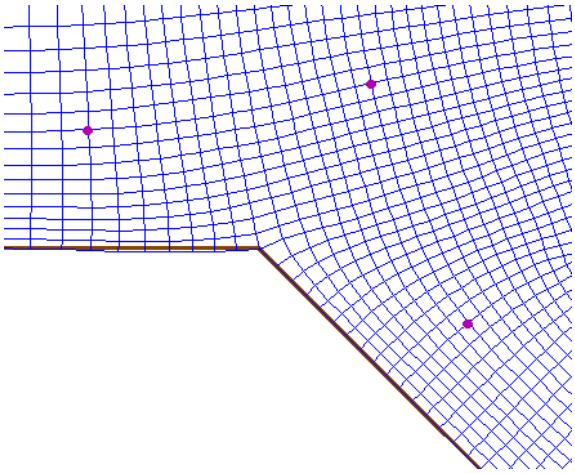


Figure 6.10: *Indicating corners for Block Orthogonalise*

6.1.10 Check other grid properties

◇ On the *Operations* menu, point to *Grid Properties* and click *Orthogonality* again

to check the result of the previous action, see Figure 6.11. Other grid properties such as grid smoothness and resolution can also be displayed. The grid should be smooth to minimise truncation errors in the finite difference scheme. Adjacent grid cell sizes should vary less than 20 percent, although locally exceptions may be acceptable.

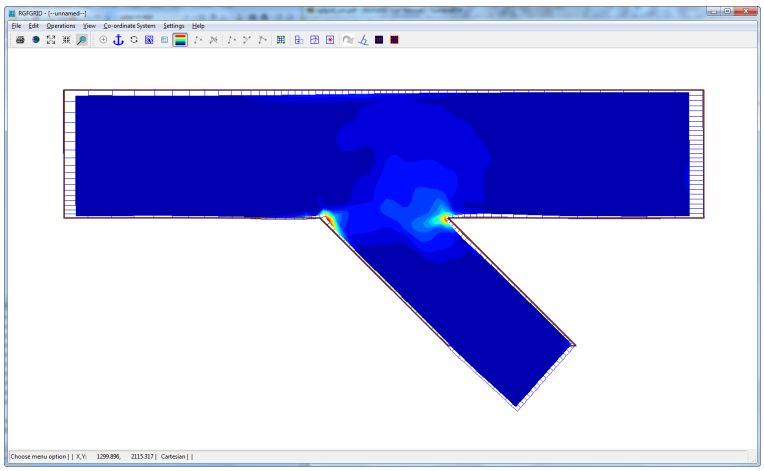


Figure 6.11: *Grid orthogonality after one block orthogonalisation operation*

6.1.11 Completion

To delete grid cells outside the land boundary:

- ◇ On the *Edit* menu, point to *Block* and click *Delete Interior*.
- ◇ Indicate a block to be deleted by clicking on opposite corners.
- ◇ Click right to activate the delete action.

You can also delete individual grid points:

- ◇ On the *Edit* menu, point to *Point* and click *Delete*.
- ◇ Press the *D*-key and click points you want to delete.

In the *Point Mode* you can also move (replace) grid points.

- ◇ First press the *R*-key, next click on a grid point, move it somewhere else, and click again.

Also in the *Point Mode*, you can add individual grid cells.

- ◇ First press the *I*-key and then click in a border cell near the concerning edge.

Do not delete grid cells outside the land boundary in earlier refinement steps, if this introduces staircase boundaries (as in the present example at the end of the harbour).

The final grid should look like Figure 6.12.

Remarks:

- ◇ Each corner point on the grid will remain fixed in the orthogonalisation procedure.
- ◇ Only internal points and points along boundaries can be shifted to improve the grid orthogonality. Creating a staircase boundary in the early stages of the iterative grid generation procedure hampers this procedure.

To shift individual grid cells:

- ◇ On the *Edit* menu, select *Point* and click *Insert*, *Move* or *Delete* or click the icons  , 

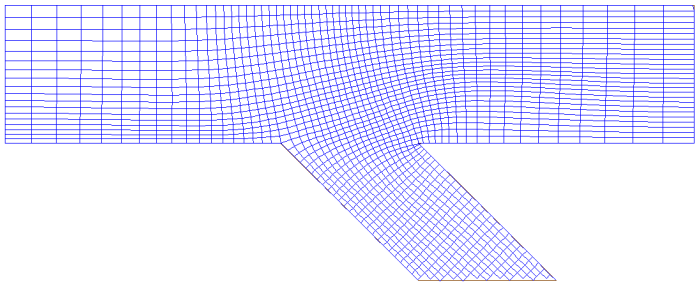


Figure 6.12: *Final result after refining, obsolete grid cells removed*

or  on the toolbar.

◇ You can also switch between the move, insert and delete actions by pressing the `I`, `R` or `D` key.

Once a mode is selected, use the left mouse to let the actions take effect. Press `Esc` to undo edit actions. In the 'spline edit mode'-mode of the program, the same keys can be used.

To exit the RGFGRID program.

◇ On the *File* menu, click *Export* → *Grid (RGFGRID)*
◇ On the *File* menu, click *Exit*.

6.2 Grid design samples

In this example we will demonstrate how to design a grid in which the shape is based on the bathymetry. This tutorial refers to the <river> directory of the RGFGRID tutorial.

◇ Start RGFGRID, and browse to the tutorial directory <river>.
◇ Open the grid file <river.grd>.
◇ Open the samples file <river.xyz>, see Figure 6.13.

◇ On the *Settings* menu, click *Orthogonalisation*:

- Set the *Design Method* to "3".
- *Depth Design, Depth vs. Slope Weight* to "0.2"

◇ Click *OK*.
◇ On the *Operations* menu, click *Orthogonalise Grid*, the result is shown in Figure 6.14.

To exit the RGFGRID program

◇ On the *File* menu, click *Export* $rightarrow$ *Grid (RGFGRID)*.
◇ Click *Exit* on the *File* menu.

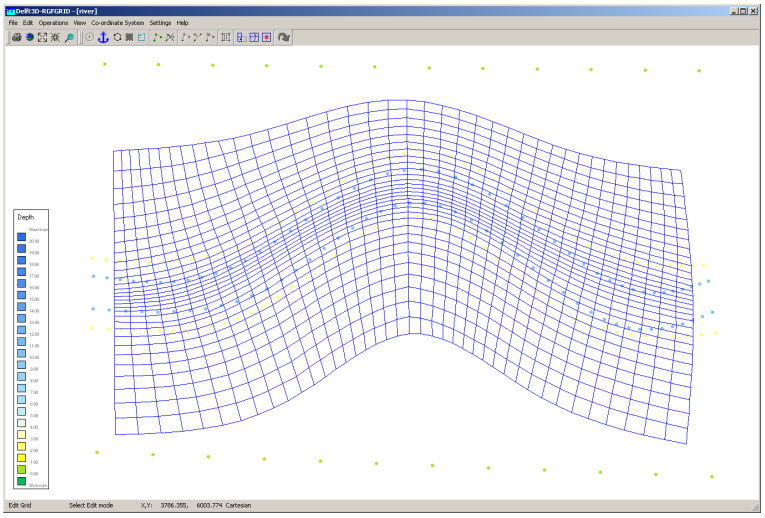


Figure 6.13: *Grid and samples for the grid design based upon bathymetry*

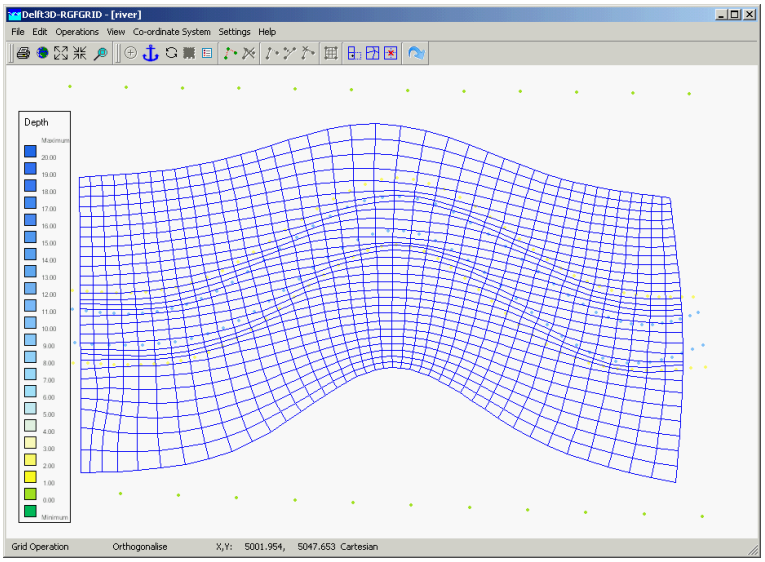


Figure 6.14: *Result grid after orthogonalisation using samples*

6.3 Paste two grids

This tutorial refers to the <Paste_Passive_Grid_to_Grid> directory of the RGFGRID tutorial.

A reason to paste two grids can be to extend an existing grid with another grid.

- ◇ Start RGFGRID, and browse to the tutorial directory <Paste_Passive_Grid_to_Grid>.
- ◇ Open the grid files <fti_02_north.grd> and <fti_02_south.grd.>
- ◇ On the *Operations* menu, click *Paste two Grids*.

to paste the two grids.

Restrictions:

- ◇ Perpendicular to the intersection line, the grid lines of both grids should be similar. Thus, refinements are not allowed.
- ◇ The grid points to paste should already be close to each other.

◇ This option only works if two grids are loaded in RGFGRID. After pasting two grids another grid can be loaded to paste.

6.4 Regular grids, irregular grids and their mutual coupling

This tutorial refers to the <dflowfm> directory of the RGFGRID tutorial.

The present section deals with four aspects of new functionalities related to the extension from curvilinear grids with unstructured, triangular grids:

1 to 'grow' a curvilinear grid (a 'regular' grid) from a geometric base line,
2 to generate unstructured, triangular grids: 'irregular' grids,
3 the coupling between regular and irregular grids,
4 the relation to existing regular grid generation options.

6.4.1 A new method to generate curvilinear grids

RGFGRID provides an improved method to generate curvilinear meshes directly from splines. In this method, a mesh is gradually developed from a base line of the channel towards the boundaries. This method requires less actions by the user and provides better orthogonality.

This approach can be illustrated as follows:

1 Load the land boundary file <scheldtharbour.ldb>.
2 Draw two cross-splines, intending to mark the inflow and outflow cross-section of the river part, through *Edit* → *Spline* → *New*.
3 Draw two additional splines, intended to loosely follow the riverbanks, in longitudinal direction.
4 Select one of the longitudinal splines and select the option *Edit* → *Spline* → *Spline to Land Boundary (new)*. The spline is now snapped to the land boundary. Repeat this action until you are satisfied with the result. For the second longitudinal spline, the actions can be repeated. The result of these actions is provided in the directory as <scheldtsplines.spl>.
5 To generate a curvilinear grid, choose *Settings* → *Grow grid from splines*. You will be able to set several settings of the operator. The upper 7 entries should be adapted into the values shown in Figure 6.15.

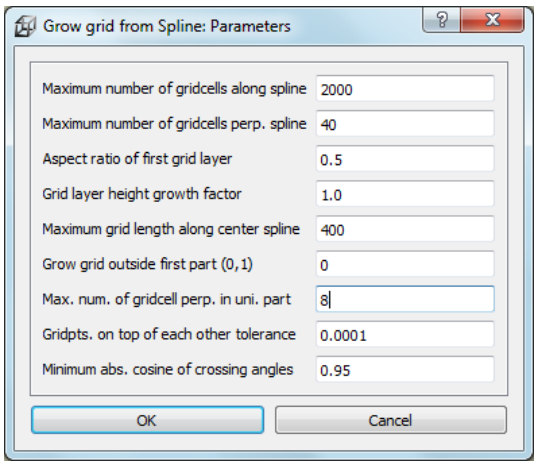


Figure 6.15: *Settings for the 'grow grid from splines' procedure.*

A brief explanation:

- ◇ Using the parameter *Max. num. of grid cells perpin uni. part*, the user can give an indication of the number of cells across the width between the longitudinal splines.
- ◇ By using the parameters *Maximum grid length along centre spline*, the user can give an indication of the length of the cells in longitudinal direction. Based on the value of the parameter *Aspect ratio of first grid layer*, the algorithm establishes a suitable mesh, under the restrictions of the prevailing maximum numbers of grid cells (first two entries).
- ◇ The option *Grid layer height growth factor* enables the user to demand for a non-equidistant mesh in cross-sectional direction. The value represents the width-ratio of two adjacent cells. Using the option *Grow grid outside first part (0/1)*, one can extend a mesh outside the longitudinal splines, for instance to capture winter bed regions of a river.

6 After entering the values of Figure 6.15, choose *Operations* → *Grow grid from splines*. This will deliver the mesh as shown in Figure 6.16.

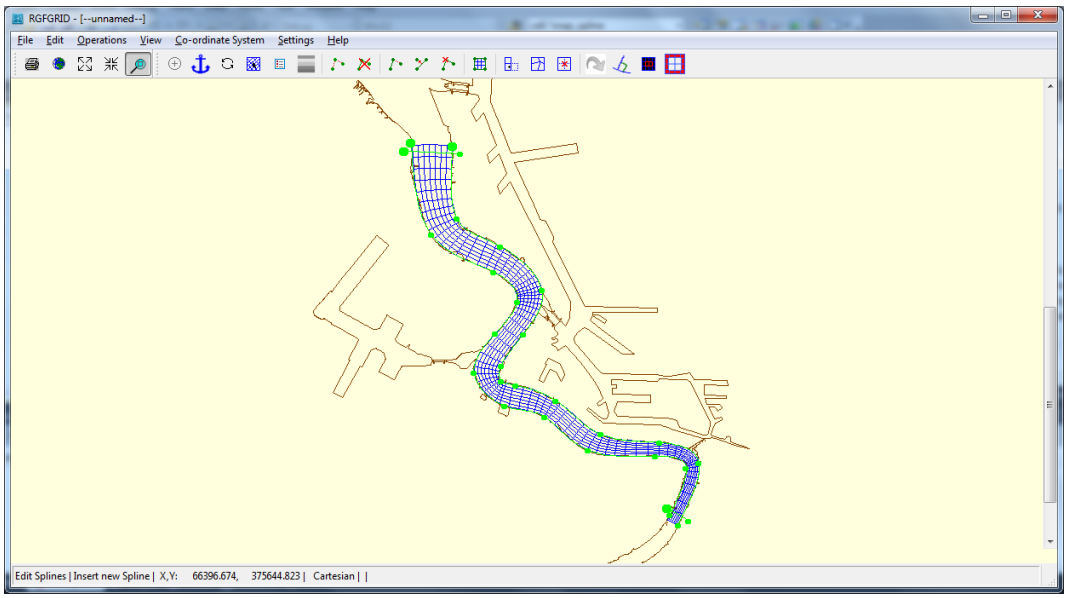


Figure 6.16: *Generated curvilinear mesh after the new 'grow grid from splines' procedure.*

7 To be able to further amend the grid, choose *Operations* → *Convert grid* → *Regular to irregular*. Strictly, the grid is now not curvilinear anymore, but unstructured.

8 Choose *View* → *Grid property* → *Coloured edge* and then *Operations* → *Grid properties* → *Orthogonality*. Now, the orthogonality of the mesh is shown. The result for the grid is also provided in the directory as <scheldtcurvi_net.nc>.

6.4.2 Irregular grids

From the previous section, a curvilinear mesh is available for the Scheldt river. The river is separated from the harbour, west of the river, by a sluice. The small area between the sluice and the Scheldt will benefit from an unstructured mesh options of RGFGRID because of its irregular geometry. This irregular geometry is meshed in this section first and afterwards connected to the existing Scheldt river mesh.

The approach is as follows:

1 Click on *Edit* → *Polygon* → *New*. The intention is to mark the area of interest (i.e. the area that should be captured by the grid) through a polygon.
2 Start drawing a polygon at a distance of the order of a grid cell away from the curvilinear mesh. Let the second point be at a relatively small distance from the first one. This distance is later used as an indication of the size of the triangular grid cells to be placed.
3 Mark the elementary locations of the area (land boundary) and place the final point again at a distance of the order of a grid cell away from the river mesh.

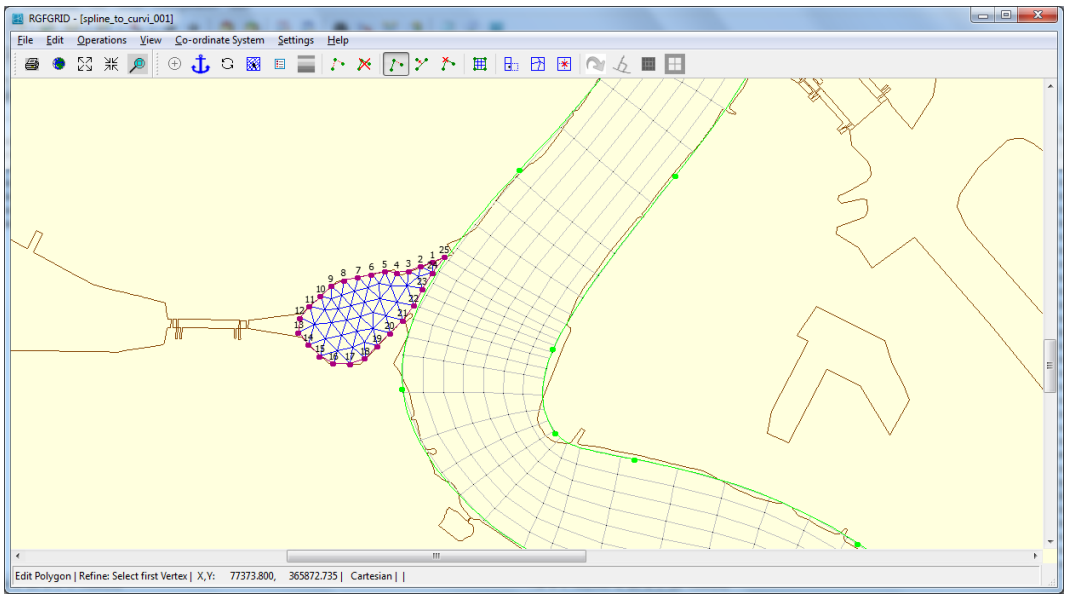


Figure 6.17: *Generated irregular grid within a polygon.*

4 Next, we choose *Edit* → *Polygon* → *Refine* and click on *two* points at the righthand-side, located close to each other, and click on the *right mouse button*. Now, the polygon is divided into a finer set of line elements. This refined polygon is also available as <scheldtpolygon.pol>. Some remarks:

   ◇ The distance between the points of the polygon is derived from the distance of the two polyline segments at both sides of the *selected* segment. The length of the polyline segments varies linearly from the segment length at the one side of the selected segment towards the segment length at the other side of the selected segment.
   ◇ You can play around to see how this works. If needed, you can add extra polyline points by choosing *Edit* → *Polygon* → *Insert point*. Choose *Edit* → *Polygon* → *Move point* if a point move would make sense.
   ◇ You can snap the refined polygon to the land boundary through *Edit* → *Edit polyline(s) within existing polygon*. The result is shown in Figure 6.17.

5 Choose *Operations* → *Create grid from polygon*. The result is shown in Figure 6.17.
6 Improve the orthogonality through *Operations* → *Orthogonalise grid*.
7 To further orthogonalise the grid by manipulating the settings, choose *Settings* → *Orthogonalisation (irregular)*, and then choosing *Operations* → *Orthogonalise grid* once again. The result for the grid is also provided in the directory as <scheldttriangle_net.nc>.

6.4.3 The coupling of regular and irregular grids

From the previous tutorial, we have ended up with two separate grids. Obviously, these two grids should properly be integrated into one single grid. Before we can couple the two grids,

we should first make sure that the typical gridsize is of the same order of magnitude for both grids at the location where the connection is to be laid. Hence, basically two operations are to be done:

- ◇ Split the grid cells in the Scheldt river grid over the full width. Hence, the gridcell size in the river will match the grid cell size of the unstructured grid.
- ◇ Merge the two grids and put connections in between of these.

The splitting can be established as follows:

1 Select the river grid through *Edit* → *Select domain* and clicking the river grid.
2 Choose *Edit* → *Grid* → *Split row or column*.
3 Select the grid lines that should be split. Start at the left boundary, and apply multiple line split operations towards the other side of the Scheldt river.
4 Try to achieve the picture shown in Figure 6.18 as regards the typical grid size in the curved area.

The merging part of the coupling schedule can be done as follows:

1 Choose *Edit* → *Allow Multi Select*. By now, you enable the option to select multiple grids.
2 Choose *Edit* → *Select domain* and click on the triangular part of the grid. As soon as you have clicked on it, both meshes are highlighted blue.
3 Merge the two seperate grids through *Operations* → *Merge grids*. Now, the grids have been merged. The result of this merging operation is provided as <scheldtmerge_net.nc>.

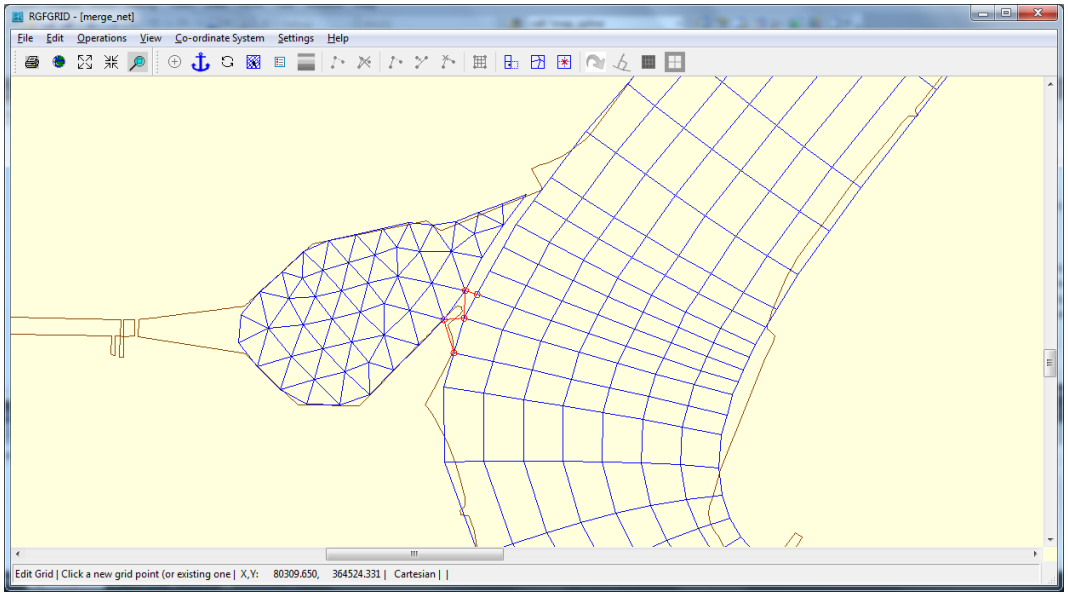


Figure 6.18: *Coupling of the two grids (regular and irregular, in blue) through manually inserting connecting grid lines (in red lines) between the two grids.*

4 As soon as the grids have been merged, new connections can be laid. Hence, choose *Edit* → *Grid* → *Edit* and then *Edit* → *Grid* → *Point insert*. Insert new gridlines in a zigzag-like style: see the red grid lines in Figure 6.18. Now, you will benefit from the (more or less) equal resolution in the river region as in the unstructured region. The integrated grid is available as <scheldtcoupling_net.nc>.

6.4.4 Relation to existing regular grid generation

The sluice area can the best be captured by a regular grid because of its rectangular shape. Thereto, you could first draw some splines in this area (see for instance Figure 6.19). Then you can establish a 3×4 grid in each block, separated by splines. To that end, choose *Operations* $\rightarrow$ *Change splines into grid*. The splines for the sluice area are available as <scheldtsluice.spl>.

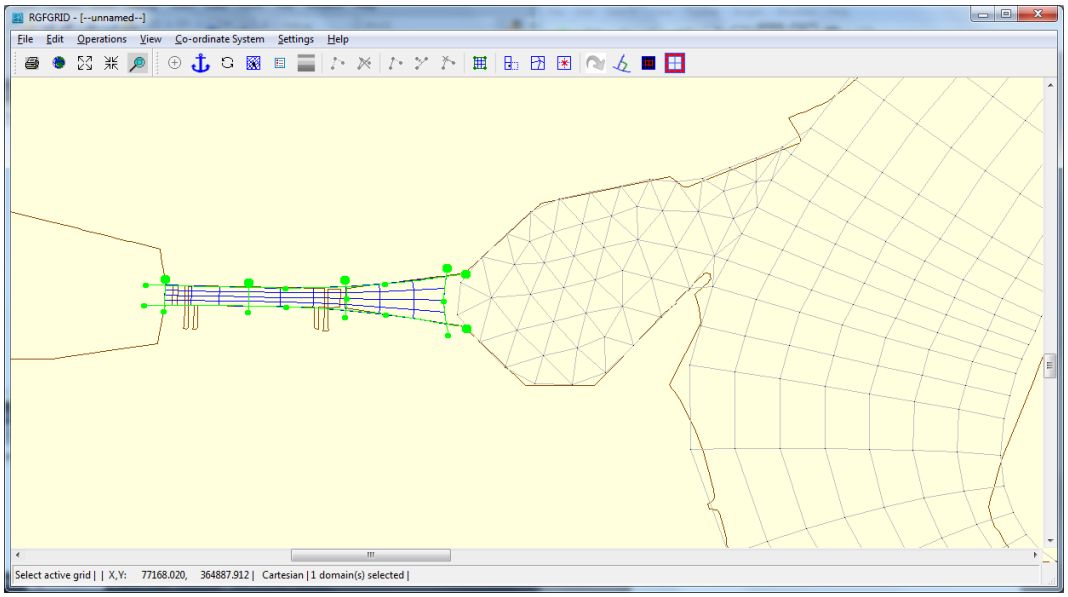


Figure 6.19: *A regular grid is suitable for the sluice area. Connections with the existing grid should further be established as well as additional orthogonalisation iterations.*

The docks of the harbour are rectangularly shaped as well. Hence, regular grids are preferred. You can also try to establish an irregular grid in this area. Thereto, draw a polygon, refine this polygon and choose *Operations* $\rightarrow$ *Create grid from polygon*.

Notice that the grid configuration as shown in Figure 6.19 needs proper connections between the sluice area and the already existing grid and, moreover, further orthogonalisation iterations before it can actually be used in computations. An example of further elaboration of the area is provided as <scheldtfinal_net.nc>.

6.5 Multi-domain grids and DD-Boundaries

This tutorial refers to the <dd_grids> directory of the RGFGRID tutorial.

The grids for a multi-domain model must satisfy the following rules:

- ◇ At sub-domain interfaces the grids should be nicely connected (no overlap and “no holes” between sub-domains), see Section 5.3.15.
- ◇ In case of horizontal, grid lines in the coarse domain should be continued in the fine sub-domain, see Figure 5.11. Thus, there should be a 1-to-N refinement, with N an integer number.
- ◇ In case of horizontal refinement it is advised to have an equidistant refinement.
- ◇ Grids must be of the same type (thus, all in Spherical co-ordinates, or all in Cartesian co-ordinates).

- ◇ The grid orientation should be the same (increasing M- or N-numbering in both sub-domains at the DD-boundary).
- ◇ No coupling of columns to rows or vice versa.
- ◇ Sub-domain interfaces should be straight lines (no stair-case interfaces).

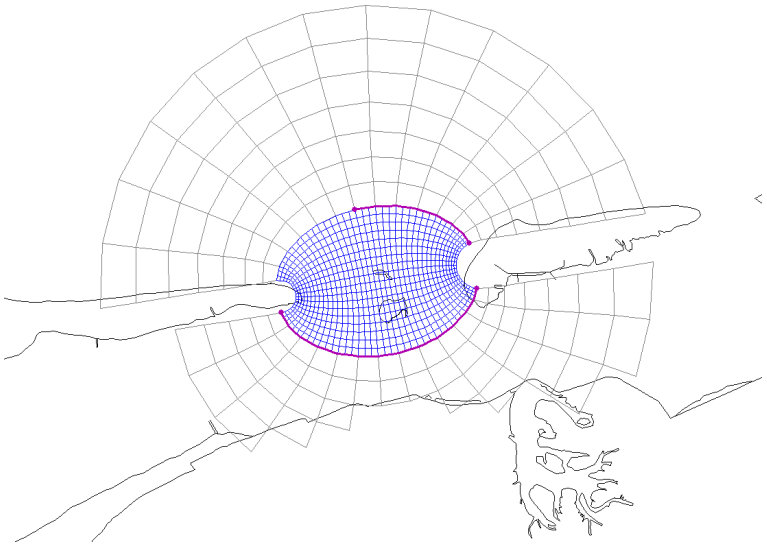


Figure 6.20: *Example of grid refinement in the horizontal direction*

RGFGRID has an option to let sub-domain interfaces coincide, see Section 5.3.15. The best moment to use this option is before refining one of the sub-domains, i.e. as long as the refinement is 1-1.

To demonstrate this functionality

- ◇ Go to directory <let_interfaces_coincide>, within the <dd_grids> directory.
- ◇ Open the grids <left_n.grd> and <right_0.grd>, zoom in on the interface between both grids.
- ◇ On the *Edit* menu, point to *DD Boundaries* → *New*.
- ◇ Click with the mouse the end points of the DD-Boundary on the interface of the active grid.
- ◇ Click *Operations* → *Attach Grids at DD Boundaries* then the whole DD-Boundary will be shifted to the interface points of the inactive grid, see Figure 6.21.

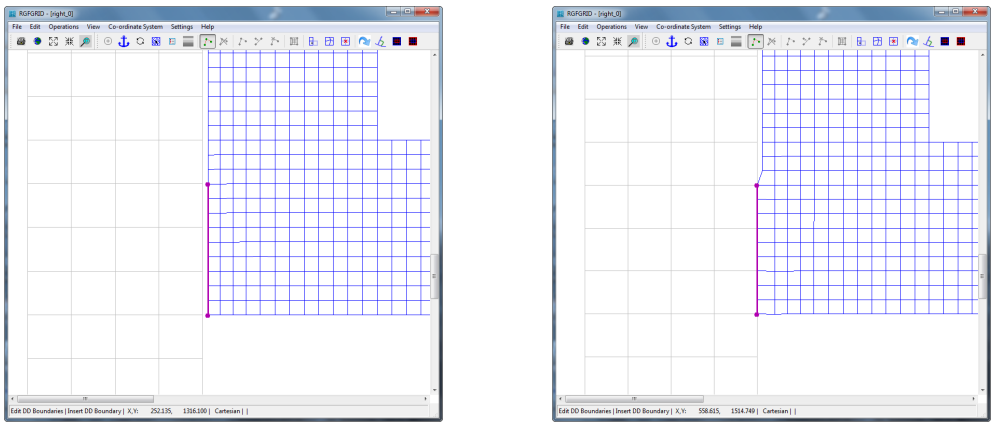
Now we have the interfaces coincide we are going to define the interfaces between the various sub-domain models.

Close and start RGFGRID in the <dd_grids> directory and load all grids into RGFGRID.

- ◇ Open the grids <bot.grd>, <left.grd>, <right.grd> and <top.grd>.

To define the DD-boundaries you select one domain as active domain, Figure 6.22:

- ◇ Make the <right.grd> the active domain
- ◇ On the *Edit* menu, point to *DD Boundaries* and click *New*.
- ◇ Specify the four DD-Boundaries as shown in Figure 6.22
- ◇ Make the <left.grd> the active domain



(a) *Before attach operation* ***(b)*** *After attach operation*

Figure 6.21: *Let interface grid points coincide*

◇ Specify the last DD-Boundaries as shown in Figure 6.22

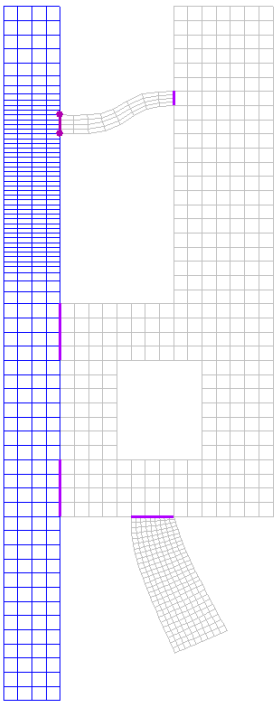


Figure 6.22: *Interfaces between sub-domains*

Now we have defined the DD-boundaries between the various domains. To gather all these information into 1 file:

◇ On the *Operations* menu, click *Compile DD Boundaries*.
◇ The **Save DD-Boundaries** dialog opens, see Figure 6.23.

◇ After pressing *OK* a message appears about the number of DD-Boundaries.
◇ Press *OK*.

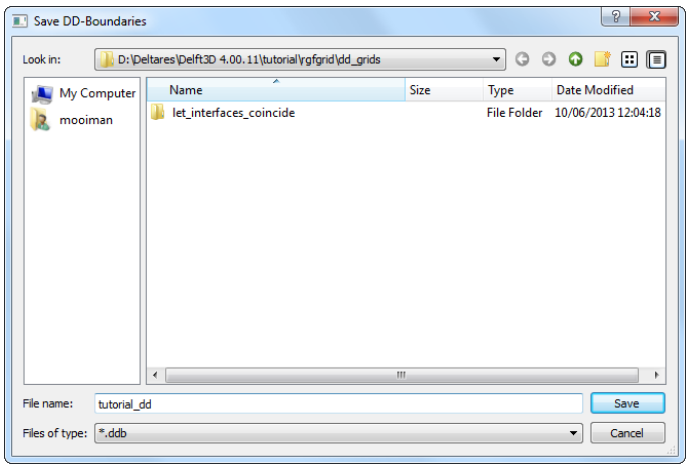


Figure 6.23: *The* ***Save DD-Boundaries*** *dialog*

The final administration will be written to a file named <tutorial_dd.ddb>. For the tutorial it looks like:

```
left.grd      5    61     5    65  top.grd       1     1     1     5
bot.grd       1    28    10    28  right.grd     6     1     9     1
left.grd      5    14     5    18  right.grd     1     1     1     5
left.grd      5    25     5    29  right.grd     1    12     1    16
top.grd       9     1     9     5  right.grd     9    30     9    31
```

Remarks:

- ◇ Before defining DD-boundaries check the orientation of each sub-domain grid.
- ◇ The orientation and order of interfaces (DD-boundaries) is free.
- ◇ It is irrelevant in which grid you define an interface, but define it once.
- ◇ If the interfacing boundaries coincide, be aware that when you orthogonalise a sub-domain grid, the grid points along these interfaces may move. To keep these points at the same place, you just re-define the DD-boundaries.

To exit the RGFGRID

- ◇ Click *Exit* on the *File* menu.

6.6 RGFGRID in the ArcMap environment

In this case you should be familiar with using co-ordinate systems in ArcMap.

ArcMap layers (e.g. shape files, SDE layers) most times have also information about the co-ordinate system (spatial reference). If not available, ArcMap marks this as unknown. On the other hand, if it is known to you, spatial reference can be added to these layers by the program ArcCatalog.

If the layers in ArcMap have a projected co-ordinate system or probably an unknown co-ordinate system, then you can use *Cartesian* in RGFGRID. It is your responsibility the co-ordinates have the unit metres. You can see the used (projected) co-ordinate system of the layers and of the data frame via the properties of the data frame. For RGFGRID the co-ordinate system of the data frame is leading. As you know, ArcMap has possibilities to set the

co-ordinate system of the data frame, while layers have different co-ordinate systems.

If you want to use spherical co-ordinates in RGFGRID while using ArcMap, the co-ordinate system of the data frame must be WGS84 (in ArcMap it has the name CGS_WGS_1984). This will be the case when all layers have this co-ordinate system.

If you are familiar with ArcMap you can have one or more layers with different co-ordinate systems and select (import) the WGS84 system for the data frame. Figure 6.24 shows the properties window of de data frame, <custom> give the co-ordinate system of the data frame.

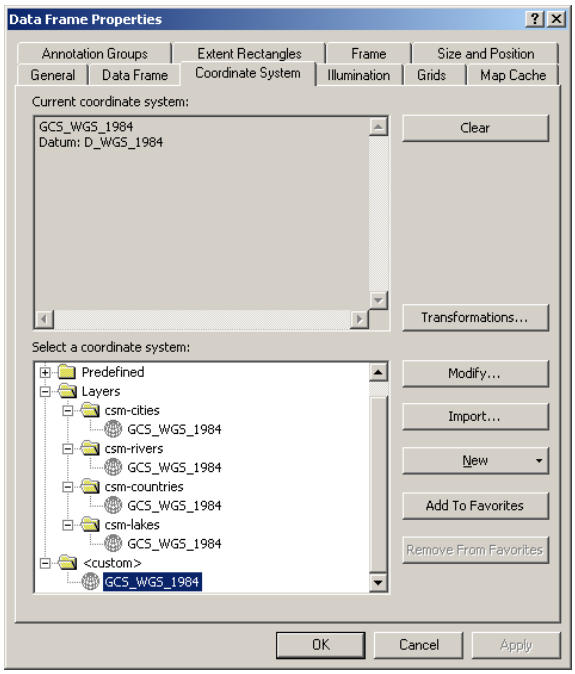


Figure 6.24: *ARC-GIS data frame properties form*

You start loading layers or an <∗.mxd > file in ArcMap. The co-ordinate system of the data frame must be as described above. You will see that ArcMap displays the values of longitude and latitude as plane co-ordinates in degrees.

Then you can start using the commands and functions of RGFGRID. The program has a menu item to change into projected co-ordinates. When using this command, the layers in ArcMap are displayed as projected and also the grid, polygons, samples of the program.

When using the menu item *View* → *Spherical Co-ordinates* → *Plane Co-ordinates*, both the layers in ArcMap and the objects of Dleft3D-RGFGRID are displayed as plane co-ordinates.

Centring the screen in stereographic mode

In the standalone version of RGFGRID, the visualisation using the stereographic projection method always uses the centre of the screen as the point where the screen touches the sphere. This is more difficult to realise when working within the GIS environment because screen handling now is governed by the GIS system. Therefore a special function has been implemented to perform this task of centring the computer screen. This function can be invoked by clicking the menu item *Operations* → *Change Centre of Projection*, see Figure 6.25. When using this command the centre of the projection is set to the centre of the screen. This action needs recalculation of the projection and a new screen refresh. By using this command

you are aware of changing the screen. When using zoom in, zoom out, pan, the centre of the projection does not change. So there is more performance and a smooth screen-refresh in this case.

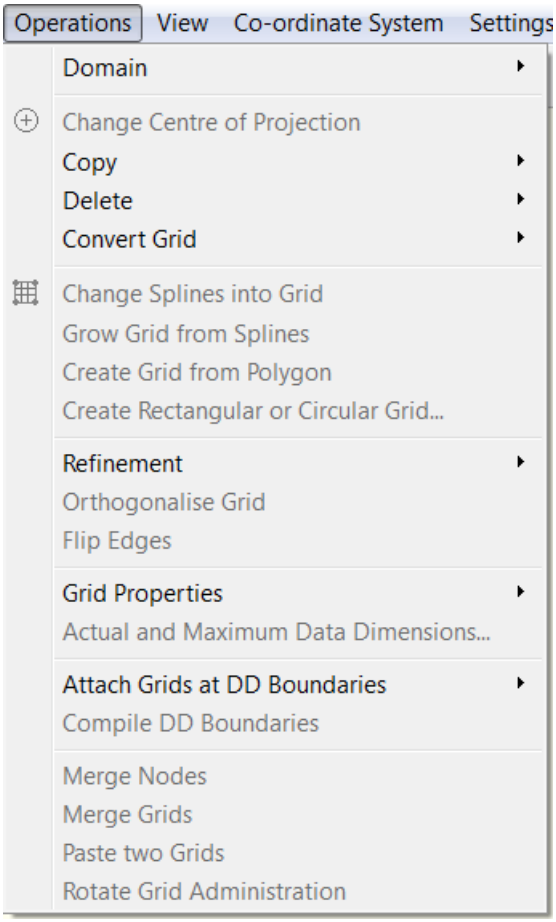


Figure 6.25: *Options on the* Operations *menu*

References

Delft3D-FLOW, 2013. *Delft3D-FLOW User Manual*. Deltares, 3.14 ed. 2

GPP, 2013. *Delft3D-GPP User Manual*. Deltares, 2.14 ed. 22

Huang, W., 2001. "Practical Aspects of Formulation and Solution of Moving Mesh Partial Differential Equations." *Journal of Computational Physics* 171: 753–775. 69, 70

QUICKPLOT, 2013. *Delft3D-QUICKPLOT User Manual*. Deltares, 2.14 ed. 22

SWAN, 2000. *SWAN Cycle III version 40.11 User Manual (not the short version)*. Delft University of Technology, Delft, The Netherlands, 0.00 ed. 2

A Files of RGFGRID

In the following sections we describe the attribute files used in RGFGRID.

For each file which can be handled by RGFGRID we give the following information:

- ◇ File contents.
- ◇ Filetype (free formatted, fix formatted or unformatted).
- ◇ Filename and extension.
- ◇ Generated by (i.e. how to generate the file).
- ◇ Restrictions on the file contents.
- ◇ Example(s).

Remarks:

- ◇ The access mode of all attribute files is sequential.
- ◇ In the examples the file content is printed in font Courier and comment (not included in the file) between curly brackets font, unless explicitly stated differently.

A.1 Delft3D project file

File contents	Domain input for a model.
Filetype	ASCII
File format	Free formatted.
Filename	<*name*.d3d>
Generated	RGFGRID, QUICKIN, D-Waq DIDO, or manually offline

Record description:

A header block containing general information and then for each domain a detailed description.

Keyword	Description
FileInformation	
FileCreatedBy	Version string of the program who generated this file the first time
FileCreationDate	Creation date and time
FileVersion	Version number of $<*$.d3d$>$ file
Geometry	
LandBoundaryName	Name of the file with the land boundaries
LandBoundaryFormat	Format of the land boundary file, possible values are: TEKAL, NETCDF or SHAPEFILE. The NetCDF file is according the 'World Vector Shoreline' format
DDBound	
FileDDBound	Name of the file with the domain decomposition boundaries

For each grid

Keyword	Description
Grid	
Type	Format of the grid file, possible values are: RGF, RGF_NETCDF, DFLOW_FM, TELEMAC
FileName	Name of grid file with the geographical co-ordinates
FlowDepth	Name of the file containing the depth values at the cell corners of the grid
Aggregation	Name of the aggregation file

Restriction:

- ◇ The maximum record length in the file is 132.

Example:

The model friesian_tidal_inlet contains three different subdomains (f01, f02, f03) and the project file has the name <friesian_tidal_inlet.d3d.>

```
[FileInformation]
  FileGeneratedBy  = Deltares, Delft3D-DIDO Version 4.04.00.11836M, Jun 21 2010, 12:09:34
  FileCreationDate = 2010-06-21, 13:35:22
  FileVersion      = 0.03
[DDBound]
  FileDDBound = f34-123.ddb
```

```
[Grid]
  Type             = RGF
  FileName         = f01.grd
  Aggregation      = f34_dd-f01.dwq
  Monitoring Areas = f34_dd-f01.dmo
[Grid]
  Type             = RGF
  FileName         = f02.grd
  Aggregation      = f34_dd-f02.dwq
  Monitoring Areas = f34_dd-f02.dmo
[Grid]
  Type             = RGF
  FileName         = f03.grd
  Aggregation      = f34_dd-f03.dwq
  Monitoring Areas = f34_dd-f03.dmo
```

A.2 Land boundary file

File contents	The co-ordinates of one or more polylines. Each polyline (piecewise linear) is written in a single block of data.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.ldb>
Generated	RGFGRID, QUICKIN, etc

Record description:

Record	Record description
	Preceding description records, starting with an asterisk ($*$), and will be ignored.
1	A non blank character string, starting in column one
2	Two integers representing the numbers of rows and number of columns for this block of data
	Two reals representing the x, y or λ, ϕ-co-ordinate

Example:

```
*
* Polyline L007
*
L007
6  2
            132400.0    549045.0
            132345.0    549030.0
            132165.0    549285.0
            131940.0    549550.0
            131820.0    549670.0
            131585.0    549520.0
*
* Polyline L008
*
L008
```

```
4  2
            131595.0    549685.0
            131750.0    549865.0
            131595.0    550025.0
            131415.0    550175.0
*
* Polyline L009
*
L009
6  2
            131595.0    549655.0
            148975.0    564595.0
            150000.0    564935.0
            152105.0    565500.0
            153150.0    566375.0
            154565.0    567735.0
```

Remark:

- ◇ In case this file is read as a polygon file then the polylines are closed by RGFGRID to get a polygon.

A.3 Sample file

File contents	The location and value of samples.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.xyz>
Generated	Manually or Offline with QUICKIN and data from digitised charts or GIS-database.

Record description:

Filetype	Record description
Free formatted	Location and sample value per row Two reals representing the x, y or λ, ϕ-coordinate and one real representing the sample value

Example:

Sample file with 12 sample values with their location (free formatted file).

```
  213813.2       603732.1       -4.053000
  214686.0       607226.1       -4.522000
  214891.7       610751.2       -5.000000
  210330.8       601424.1       -2.169000
  211798.0       604444.8       -2.499000
  212460.0       607475.7       -2.760000
  212436.9       610362.5       -2.865000
  185535.4       606607.9        1.360000
  186353.0       603789.4        1.122000
  187959.2       601197.6       0.9050000
  190193.0       599101.5       0.7050000
  208578.7       602513.7      -0.7990000
```

A.4 Spline file

File contents	The co-ordinates of one or more polygons. Each polygon is written in a single block of data
Filetype	ASCII
File format	Free formatted
Filename	<*name*.spl>
Generated	Delft3D-RGFRID

Record description:

Record	Record description
	Preceding description records, starting with an asterisk ($*$), and will be ignored.
1	Character string of at least 1 character.
2	Two integers representing the numbers of rows and number of columns for this block of data
	Two reals representing the x, y or λ, ϕ-coordinate

Example:

```
*
* Deltares, \DRGFGRID\ Version 4.16.01.4887, Oct 18 2008, 13:26:48
* File creation date: 2008-10-19, 13:33:05
*
* Coordinate System = Cartesian
*
S001
   6   2
 -1.1520000E+02   9.9630000E+02
  1.2911200E+03   9.9878100E+02
  2.2075800E+03   1.0299500E+03
  3.0180600E+03   1.3105000E+03
  4.1090800E+03   1.3479100E+03
  5.1315300E+03   1.3354400E+03
S002
   2   2
  3.4607000E+03  -6.0347500E+02
  4.0405100E+03   5.7377700E+01
```

A.5 Polygon file

File contents	The co-ordinates of one or more polygons. Each polygon is written in a single block of data
Filetype	ASCII
File format	Free formatted
Filename	<*name*.pol>
Generated	RGFGRID, QUICKIN, D-Waq DIDO, etc

Record description:

The file may contain one or more polygons. For every polygon the file should contain a line indicating the name of the polygon, followed by a line indicating the number of points making up the polygon and the number of coordinates, i.e. 2, finally followed by the coordinate data.

Record	Record description
	Preceding description records, starting with an asterisk ($*$), and will be ignored.
1	A non blank character string, starting in column one
2	Two integers representing the numbers of rows and number of columns for this block of data
	Two reals representing the x, y or λ, ϕ-coordinate

Restriction:

- ◇ The first record and the last record in the block should be the same

Example:

```
*
* Deltares, Delft3D-DIDO Version 3.39.01.4423:4459, Sep 25 2008, 20:10:54
* 2008-09-25, 22:11:08
*
Observation Area 001
       5   2
   1.8768018E+05   6.1708738E+05
   1.8996981E+05   6.1001035E+05
   1.9746314E+05   6.1266423E+05
   1.9480925E+05   6.1838830E+05
   1.8768018E+05   6.1708738E+05
Observation Area 002
       5   2
   2.0011703E+05   6.1818015E+05
   1.9819166E+05   6.1063479E+05
   2.0568498E+05   6.0870942E+05
   2.0797461E+05   6.1599460E+05
   2.0011703E+05   6.1818015E+05
Observation Area 003
       5   2
   1.9340425E+05   6.1396516E+05
   2.0183425E+05   6.1365294E+05
   1.9944054E+05   6.0558720E+05
   1.9522555E+05   6.0595146E+05
   1.9340425E+05   6.1396516E+05
```

A.6 Orthogonal curvilinear grid file

File contents	The co-ordinates of the orthogonal curvilinear grid at the depth points.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.grd>
Generated	RGFGRID

Record description:

Record	Record description
	Preceding description records, starting with an asterisk ($*$), will be ignored.
1	Record with `Co-ordinate System = Cartesian` or value `Spherical`
2	The number of grid *points* in m- and n-direction (2 integers).
3	Three real values (not used).
4 to K+3	A label and record number, the x-component of the world co-ordinates of all points in m-direction, starting with row 1 to row $nmax$, with as many continuation records as required by $mmax$ and the number of co-ordinates per record. The label and record number are suppressed on the continuation lines. This set of records is repeated for each row until $n = nmax$.
K+4 to 2K+3	A similar set of records for the y-component of the world co-ordinates.

K is the number of records to specify for all grid points a set of x- or y-co-ordinates.

Restrictions:

- ◇ The grid must be orthogonal.
- ◇ Input items in a record are separated by one or more blanks.

Example:

```
*
* Deltares, Delft3D-RGFGRID  Version 4.16.01.4531, Sep 30 2008, 23:32:27
* File creation date: 2008-10-01, 23:19:22
*
Coordinate System = Cartesian
       9       7
 0 0 0
 Eta=    1   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
             5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=    2   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
             5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=    3   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
             5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
 Eta=    4   0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
```

```
                5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
Eta=     5      0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
                5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
Eta=     6      0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
                5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
Eta=     7      0.00000000000000000E+00   1.00000000000000000E+02   2.000000...
                5.00000000000000000E+02   6.00000000000000000E+02   7.000000...
Eta=     1      1.00000000000000000E+02   1.00000000000000000E+02   1.000000...
                1.00000000000000000E+02   1.00000000000000000E+02   1.000000...
Eta=     2      2.00000000000000000E+02   2.00000000000000000E+02   2.000000...
                2.00000000000000000E+02   2.00000000000000000E+02   2.000000...
Eta=     3      3.00000000000000000E+02   3.00000000000000000E+02   3.000000...
                3.00000000000000000E+02   3.00000000000000000E+02   3.000000...
Eta=     4      4.00000000000000000E+02   4.00000000000000000E+02   4.000000...
                4.00000000000000000E+02   4.00000000000000000E+02   4.000000...
Eta=     5      5.00000000000000000E+02   5.00000000000000000E+02   5.000000...
                5.00000000000000000E+02   5.00000000000000000E+02   5.000000...
Eta=     6      6.00000000000000000E+02   6.00000000000000000E+02   6.000000...
                6.00000000000000000E+02   6.00000000000000000E+02   6.000000...
Eta=     7      7.00000000000000000E+02   7.00000000000000000E+02   7.000000...
                7.00000000000000000E+02   7.00000000000000000E+02   7.000000...
```

A.7 Grid enclosure file

File contents	The indices of the external computational grid enclosure(s) and optionally one or more internal computational grid enclosures that outlines the active computational points in a Delft3D-FLOW computation. The file is strongly related to the curvilinear grid file.
Filetype	ASCII
File format	Free formatted
Filename	<*name*.enc>
Generated	RGFGRID

Record description:

Record	Record description
All	One pair of M and N indices representing the grid co-ordinates where a line segment of the computational grid enclosure (polygon) changes direction.

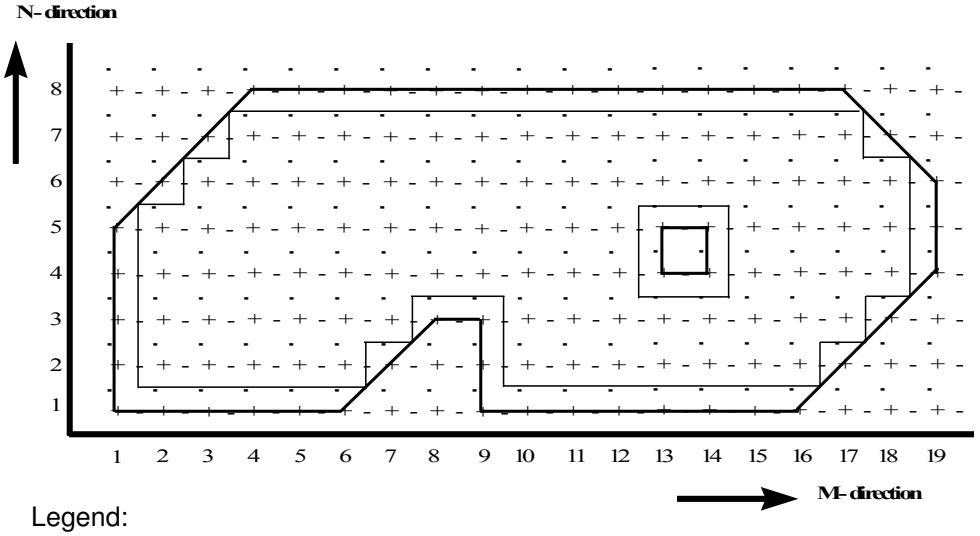
Restrictions:

- ◇ A polygon must be closed. The first point of the polygon is repeated as last point.
- ◇ A line segment may not intersect or touch any other line segment.
- ◇ The angle formed by consecutive line segments (measured counter clock-wise) can have a value of: 45, 90, 135, 225, 270 or 315 degrees, but not 0, 180 and 360 degrees.
- ◇ In a row or column there should be at least two active computational grid cells.
- ◇ Input items in a record are separated by one or more blanks.

Example:

Model area with (one) external and one internal polygon, see Figure A.1.

```
1  1   begin external polygon
```



Legend:

+	water level point
|	v-velocity point
−	u-velocity point
Full thick line	grid enclosure and (for the external polygon only) location of water level open boundaries.
Full thin line	location for velocity or discharge open boundaries.

Figure A.1: *Example of computational grid enclosures*

```
 6  1
 8  3
 9  3
 9  1
16  1
19  4
19  6
17  8
 4  8
 1  5
 1  1   end external polygon
13  4   begin internal polygon
14  4
14  5
13  5
13  4   end internal polygon
```

A.8 Annotation file

File contents	File with x and y co-ordinates, string and rgb-colour.
Filetype	ASCII
File format	Free formatted.
Filename	<*name*.txt>
Generated	manually offline

Record description:

Record	Record Description
1	Records starting with a $*$ are comment lines Character string to define the datablock (nonblank)
2	Number of rows
3–N	real, real, string, integer: geographical co-ordinates (2 reals), text between quotes which need to be plotted (string) and rgb-colour (integer; $= 256 * 256 * r + 256 * g + b$)

Restriction:

- ◇ The maximum record length in the file is 132.

Example:

```
*
BL01
8 4
10.0  10.0   'string-01'  6553625
10.0  20.0   'string-02'  9830425
20.0  10.0   'string-03' 13120000
20.0  20.0   'string-04' 16724480
30.0  10.0   'string-05'    38425
30.0  20.0   'string-06'    65305
40.0  10.0   'string-07'      255
40.0  20.0   'string-08'        0
```

A.9 DD Boundary file

File contents	Domain decomposition boundaries connecting two grids for the prescribed indices.
Filetype	ASCII
File format	Fix formatted.
Filename	<*name*.ddb>
Generated	RGFGRID, or manually offline

Record description:

Record	Record Description
N	Name of the first grid, followed by four integers indicating the gridline on which the boundary lies, followed by the name of the second grid and four integers indicating the gridline on which the boundary lies.

Restrictions:

- ◇ No space allowed in grid filename.
- ◇ The maximum record length in the file is 132.

Example:

In the following example 4 sub-domains exist. Domain d01_ns is coupled to oa1_ns, ob1_ns and oc1_ns. Furthermore oa1_ns is coupled to ob1_ns, and ob1_ns to oc1_ns.

```
d01_ns.grd            5   16    5    1  oa1_ns.grd           28   35   28   20
d01_ns.grd          245    1    5    1  ob1_ns.grd           17   21    1   21
d01_ns.grd          245   52  245    1  oc1_ns.grd            1   44    1   27
ob1_ns.grd            1    4    1   21  oa1_ns.grd           28    3   28   20
ob1_ns.grd           17    4   17   21  oc1_ns.grd            1   10    1   27
```

A.10 Colour scheme file

File contents	The colour scheme
Filetype	ASCII
File format	Free formatted
Filename	<*name*.clr> or <*name*.clrmap>
Generated	manually

Record description:

Record	Record description
1	COLORMAP
2	NAME=name
3	SPACE=RGB, RGB is the only allowed space for this program
4 – N	one real and three integers.

The first column represent the relative distribution of the defined colours in column 2–4 (representing the RGB values).

Example:

```
COLORMAP
NAME=copper
SPACE=RGB
0.0000   0   0   0
0.8000 255 159 101
```

```
1.0000 255 199 127
```

A.11 Settings file

File contents	Settings of the program
Filetype	ASCII
File format	Fix formatted
Filename	<*name*.ini>
Generated	By the program

Record description:

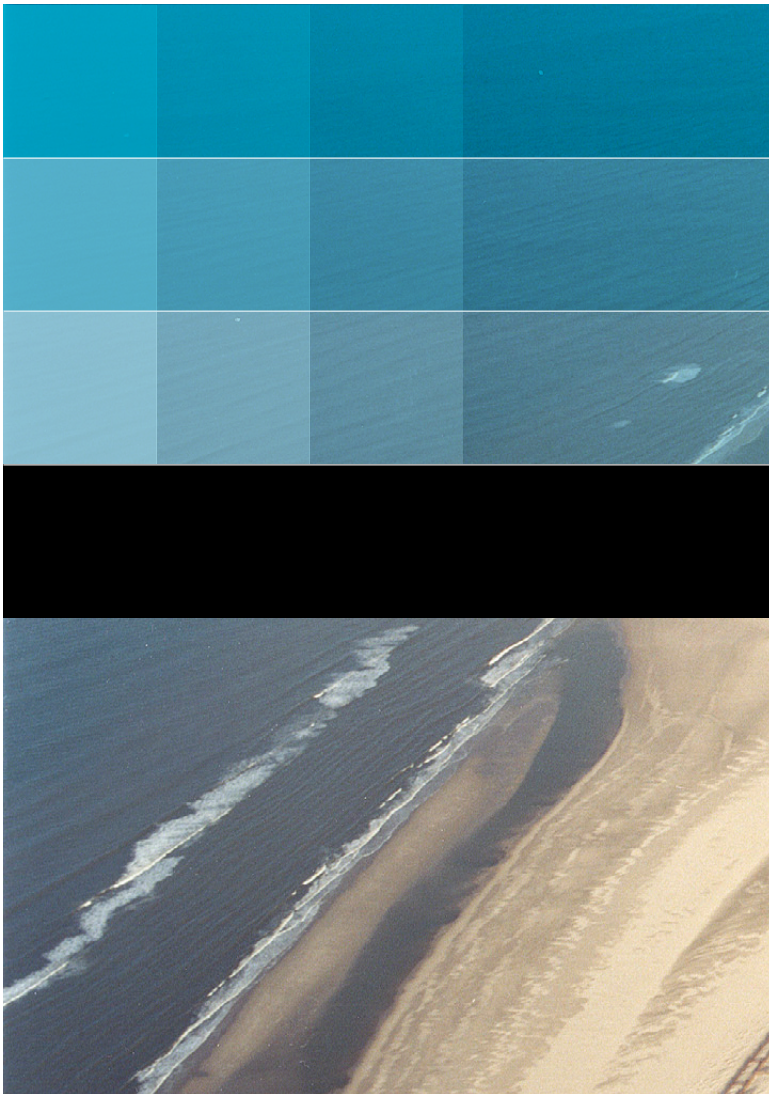
Record	Record description
FileInformation	
FileCreatedBy	RGFGRID version number
FileCreationDate	creation date and time
FileVersion	version number of <∗.ini> file
RGFParameter	
name	integer value
TextSettings	
name	value (integer or real)
RGFsettings	
name	value (integer or real)
DepthDesign	
name	value (integer or real)
Colours	
name	RGB value (3 integers) line width dots sizes

Example:

```
[FileInformation]
  FileGeneratedBy  = Deltares, Delft3D-RGFGRID Version 4.20.00.11763:11790M, Jun 16 2010, 14:21:46
  FileCreationDate = 2010-06-16, 14:23:25
  FileVersion      = 0.02
[RGFParameter]
  AutoscaleLegend                 =   1
  XCoorLegend                     =  16
```

```
  YCoorLegend                     =  20
[TextSettings]
  Line1                           =
  Line2                           =
  Line3                           =
  Fontsize                        =   3.0000000000000000E+00
  Xposition                       =   0.0000000000000000E+00
  Yposition                       =   0.0000000000000000E+00
  FontsizeTimeDate                =   3.0000000000000000E+00
  XposTimeDate                    =   0.0000000000000000E+00
  YposTimeDate                    =   3.0000000000000000E+00
[RGFSettings]
  MGridCells                      =  50
  NGridCells                      =  50
  DeltaX                          =   1.0000000000000000E+02
  DeltaY                          =   1.0000000000000000E+02
  OriginX                         =   0.0000000000000000E+00
  OriginY                         =   1.0000000000000000E+02
  RotationLeft                    =   0.0000000000000000E+00
  RadiusCurvatureM                =   0.0000000000000000E+00
  MFraction                       =   1.0000000000000000E+00
  MaximumSizeUniformMSize         =   1.0000000000000000E+00
  NFraction                       =   1.0000000000000000E+00
  MaximumSizeUniformNSize         =   1.0000000000000000E+00
  SphereMGridCells                =  50
  SphereNGridCells                =  50
  SphereDeltaX                    =   1.0000000000000002E-02
  SphereDeltaY                    =   1.0000000000000002E-02
  SphereOriginX                   =   4.38082999999999956E+00
  SphereOriginY                   =   5.19858300000000000E+01
  SphereRotationLeft              =   0.0000000000000000E+00
  SphereRadiusCurvatureM          =   0.0000000000000000E+00
  SphereMFraction                 =   1.0000000000000000E+00
  SphereMaximumSizeUniformMSize   =   1.0000000000000000E+00
  SphereNFraction                 =   1.0000000000000000E+00
  SphereMaximumSizeUniformNSize   =   1.0000000000000000E+00
  StayOnStartupDirectory          =   0
  MRefinementFactor               =   3
  NRefinementFactor               =   3
  NrSmoothingIterations           =  20
  SmoothingParameter              =   2.0000000000000011E-01
  AttractionRepulsionParameter    =   1.0000000000000006E-01
  ActiveInactivePaste             =   5.0000000000000000E-01
  LineOrSplineRepresentation      =   1.0000000000000000E+00
  EquidistantSmoothInterpolation  =   1.0000000000000000E+00
  IncreaseFactorLineMirror        =   1.0000000000000000E+00
  IterationsAttractionParameter   =   3
  IterationsBoundary              =  15
  IterationsInnerArea             =  25
  InfluenceOriginalGridShape      =   1.0000000000000000E+00
  PositionBoundaryPoints          =   1.0000000000000000E+00
  DesignMethod                    =   1
[DepthDesign]
  DepthDesignSizeRatioM           =   2.0000000000000011E-01
  DepthDesignSizeRatioN           =   2.0000000000000011E-01
  DepthDesignDepthVsSlope         =   1.0000000000000000E+00
  DepthDesignNrSmoothingIterations =  10
  DepthDesignSmoothingFactor      =   1.0000000000000006E-01
  DepthDesignFieldVsLineWeightM   =   0.0000000000000000E+00
  DepthDesignFieldVsLineWeightN   =   0.0000000000000000E+00
[Colours]
  ColourBackground                = 255 255 200
```

```
  LegendColourBackground          = 255 255 255
  lineColourText                  = 000 255 255
  lineColourLandBoundary          = 132 066 000
  fillColourLandBoundary          = 255 182 108
  lineColourSplines               = 000 255 000
  lineColourPolyline              = 000 255 000
  fillColourPolyline              = 000 255 000
  lineColourPolygon               = 170 000 127
  lineColourActiveGrid            = 000 000 255
  lineColourPreviousGrid          = 211 000 000
  lineColourActiveCmpBnd          = 005 005 005
  lineColourActiveOpenBnd         = 000 000 255
  lineColourActiveDDBnd           = 176 000 176
  lineColourGrid                  = 192 192 192
  lineColourCmpBnd                = 050 050 050
  lineColourOpenBnd               = 000 000 150
  lineColourDDBnd                 = 176 000 255
[Width]
  lineWidthLandBoundary           =   1
  lineWidthPolyline               =   1
  lineWidthPolygon                =   1
  DotSizeSamples                  =   2
  lineWidthActiveGrid             =   1
  lineWidthActiveCmpBnd           =   1
  lineWidthActiveOpenBnd          =   1
  lineWidthActiveDDBnd            =   3
  lineWidthGrid                   =   1
  lineWidthCmpBnd                 =   1
  lineWidthOpenBnd                =   1
  lineWidthDDBnd                  =   3
[Caches]
  splines                         =   0
  rest                            =   0
  polygons                        =   0
  polylines                       =   0
  gridprop                        =   0
  gridadm                         =   0
  gridprev                        =   2
  cmpbound                        =   1
  openbound                       =   1
  ddbound                         =   1
  actgrid                         =   2
  inactgrid                       =   3
  landboundary                    =   4
  samples                         =   5
```



Deltares systems

PO Box 177
2600 MH Delft
Rotterdamseweg 185
2629 HD Delft
The Netherlands

+31 (0)88 335 81 88
sales@deltaressystems.nl
www.deltaressystems.nl